Proyecto Fin de Carrera

Grado en Ingeniería de las Tecnologías Industriales

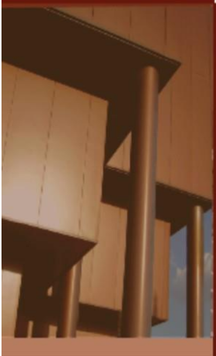
Intensificación en Mecánica de Máquinas

Fatigue strength estimations of notched specimens with Ansys.

Estimaciones de resistencia a la fatiga de probetas entalladas con Ansys.

Autor: Claudia Caus Regidor

Tutor: Carmen Madrigal Sánchez y Víctor Manuel Chaves



Dpto. Ingeniería Mecánica y Fabricación
Escuela Técnica Superior de Ingeniería
Universidad de Sevilla

Proyecto Fin de Carrera

Grado en Ingeniería de las Tecnologías Industriales

Intensificación en Mecánica de Máquinas

Fatigue strength estimations of notched specimens with Ansys.

Estimaciones de resistencia a la fatiga de probetas entalladas con Ansys.

Autor:

Claudia Caus Regidor

Tutor:

Carmen Madrigal Sánchez

Víctor Manuel Chaves Repiso

Dep. Ingeniería Mecánica y Fabricación

Escuela Técnica Superior de Ingeniería

Universidad de Sevilla

Sevilla, 2023

Acknowledgements

If I would thank everyone I should, this would be as long as the thesis; therefore, I will try to keep it as short as possible.

First, I would like to thank the "Escuela Técnica Superior de Ingeniería" and all the amazing professor I have had these years. I have learnt and developed myself more that I could imagine, especially to those who always motivated me and made me love what I study every day. I would like to express particular thanks to Carmen and Víctor for their constant support, for everything they have taught me and for always being available when I needed it.

Also, my university experience had not been the same without ESTIEM, this association has built me as a person and given me friends for the rest of my life.

Of course, I want to thank my parents for their support and for bringing me the possibility to pursuit my study and career goals.

Lastly, I would like to thank my very close friends, who have always been there for me for years. I could write many names here, but I would like to highlight three of them who have been my daily support throughout my studies. Irene, who has been next to me since the very first day of university and she is the best I have gained from it. Víctor, my library and car buddy since high school and one of my closest friends. Claudia, practically my sister since we were ten years old and who knows me better than anyone.

Claudia Caus Regidor

Student of Industrial Technologies Engineering Bachelor

Sevilla, 2023

ç

Abstract

Nowadays the fatigue of materials is one of the most studied fields in mechanical engineering, as it is one of the most usual causes of failure. However, it is a complicated area of study, where no final theory has been found that englobes all cases, but depending on the conditions, different paths and tools should be chosen to reach the right solutions. Besides that, in order to obtain results in this field, the use of numerical calculation software is necessary in the majority of cases.

Therefore, this project presents a study of the accuracy of the solutions obtained with Ansys and a comparison between the interfaces of this software: APDL Ansys Mechanical Product Launcher and Workbench, respectively, applying the theory of critical distances of D. Taylor and the fatigue tool. As can be expected, each of the tools has its advantages and disadvantages and, depending on the purposes, it is more convenient to choose one or the other path.

Also, the fatigue tool of Workbench is relatively new, and it is difficult to imagine how a program can run fatigue predictions automatically, which raises another question, which this study aims to answer: How does this tool work and how reliable is it?

Resumen

Hoy en día la fatiga de materiales es uno de los campos más estudiados en ingeniería mecánica, ya que es una de las causas más habituales de fallo. Sin embargo, se trata de un área de estudio complicada, en la que no se ha encontrado una teoría definitiva que englobe todos los casos, sino que, dependiendo de las condiciones, se deben elegir diferentes caminos y herramientas para llegar a las soluciones adecuadas. Además, para obtener resultados en este campo, en la mayoría de los casos es necesario el uso de software de cálculo numérico.

Por ello, este proyecto presenta un estudio de la precisión de las soluciones obtenidas con Ansys y una comparación entre las interfaces de este software: APDL Ansys Mechanical Product Launcher y Workbench, aplicando respectivamente la teoría de distancias críticas de D. Taylor y la herramienta de fatiga. Como es de esperar, cada una de las herramientas tiene sus ventajas y desventajas y, dependiendo de los propósitos, es más conveniente elegir uno u otro camino.

Además, la herramienta de fatiga de Workbench es relativamente nueva, y es difícil imaginar cómo un programa puede ejecutar predicciones de fatiga automáticamente, lo que plantea otra pregunta, a la que este estudio pretende dar respuesta: ¿Cómo funciona esta herramienta y hasta qué punto es fiable?

Index

Acknowledgements .... **v**

Abstract .... **viii**

Resumen .... **ix**

Index .... **xi**

Index of Figures .... **xiii**

1. Introduction **15**

1.1. Definition of Fatigue of Materials *15*

1.2. History of Fatigue Research *16*

2. Fatigue Theories Overview **18**

2.1. Stress-Strain Curves *18*

2.2. S-N Method *22*

2.2.1. S-N Curves 23

2.3. Fracture Mechanics *26*

2.4. The Failure of Notched Specimens *31*

2.5. Finite Element Analysis *32*

2.6. Concluding Remarks *34*

3. The Theory of Critical Distances **35**

3.1. Definition *35*

3.2. Description and Applications *35*

3.3. Point Method *37*

4. Point Method in APDL Product Launcher **42**

4.1. Mechanical APDL Product Launcher *42*

4.2. Point Method Implementation to APDL *43*
4.3. Procedure of Calculations *48*

5. Implementation of the Fatigue Tool of Workbench 52
5.1. New Project Creation on Workbench *53*
5.2. Fatigue Tool *58*

6. Comparison of Results Obtained with Each Method 71

7. Conclusions 74

8. Conclusiones 76

References 78

Appendix: APDL Codes 80

INDEX OF FIGURES

Figure 1: Example of a stress-strain curve obtained with the tensile test. 20

Figure 2: Ultimate tensile strength on the stress-strain curve 21

Figure 3: Stress variables for constant amplitude cyclic stress..................................... 23

Figure 4: Theoretical S-N curve 24

Figure 5: Presentation of the test results using a Wöhler diagram (S-log N) [2]....... 24

Figure 6: Example of a Wöhler diagram for high and cyclic fatigue 25

Figure 7: Variation of the S-N curve due to the mean stress 26

Figure 8: Typical fatigue crack growth rate curves [14].. 30

Figure 9: Centre hole plane 36

Figure 10: V-notch plane.. 37

Figure 11: Representation of the PM on the centre hole plane.................................... 38

Figure 12: Representation of the PM on the V-notch plane 39

Figure 13: Symmetry of the centre hole plane 44

Figure 14: Symmetry of the V-notch plane.. 45

Figure 15: Key points of the centre hole plane.. 46

Figure 16: Key points of the V-notch plane 46

Figure 17: Lines of the centre hole plane.. 47

Figure 18: Lines of the V-notch plane 47

Figure 19: Calculation of the accuracy of the results 50

Figure 20: Variation of Kf depending on the radius.. 51

Figure 21: Relation between Kf and Kt depending on the radius. 51

Figure 22: Workbench Static Structure 53

Figure 23: S-N curve of the used stainless steel [3] 54
Figure 24: Workbench Engineering Data 54
Figure 25: Workbench New DesignModeler Geometry 55
Figure 26: Automatic mesh 56
Figure 27: Mesh after the refine. 56
Figure 28: Representation of the problem in Workbench 57
Figure 29. Bilinear S-N curve 59
Figure 30. Linear S-N curve 59
Figure 31. Tabular option 60
Figure 32: S-N curve 65
Figure 33: Fatigue Tool Life plot. Zoom of the circular notch 66
Figure 34: Fatigue Tool Damage plot. Zoom of the circular notch 66
Figure 35: Fatigue Tool Safety Factor plot. Zoom of the circular notch 67
Figure 36: Fatigue Tool Biaxiality Indication plot. Zoom of the circular notch 68
Figure 37: Fatigue Tool Equivalent Alternating Stress plot (equivalent of von Mises). Zoom of the circular notch 69

1. INTRODUCTION

1.1. Definition of Fatigue of Materials

Fatigue of materials refers to the process by which a material fails when subjected to cyclic loading, even if the magnitude of the loads is below the ultimate strength of the material. This failure occurs over time due to repeated application of stresses, leading to cracks and ultimately, the failure of the material. This deterioration of the material is affected in almost all cases by factors other than the applied load, such as temperature, environment, geometry of the specimen, etc. Fatigue affects many families of materials, like metals, plastics, and composites and it can occur in a variety of applications, including mechanical, aerospace, automotive, and biomedical engineering, among others.

The process of fatigue failure typically involves three stages: crack initiation, crack propagation, and final failure. During crack initiation, microscopic cracks or defects form on the surface of the material due to the repeated application of cyclic loading. These cracks then propagate through the material until they reach a critical size, at which point the material fails.

Fatigue of materials is a wide and not highly defined area of engineering, i.e., various techniques and theories to prevent this kind of failure have been studied in the last decades; however, none of them could be presented as a definitive theory. Most of these theories are based on empirical data and supported by the use of finite element analysis (FEA) and other simulation tools, since the calculations can only be obtained analytically in a small number of cases.

In machinery, movement is essential and consequently so are cyclic stresses and vibrations, which lead to fatigue damage. For this reason, the study of the fatigue of materials is such an important field in mechanical and aerospace engineering, thereby conditioning in numerous cases the choice of material, the design, or the manufacturing process.

1.2. History of Fatigue Research

The history of fatigue can be traced back to the mid-19th century when engineers first observed the failure of railroad axles after prolonged use, even under relatively low loads. These observations sparked curiosity and raised questions about the underlying mechanisms behind this type of failure. In 1854, the German engineer August Wöhler conducted pioneering experiments to systematically study fatigue. He subjected various materials, including iron and steel, to repeated loading and observed their behaviour. Wöhler's experiments revealed that materials exhibited a limited fatigue life and that the number of cycles to failure decreased as the applied stress increased. This laid the foundation for further investigations into fatigue phenomena.

In the late 19th century, researchers expanded on Wöhler's work and introduced the concept of an "endurance limit". They observed that some materials had a stress level below which they could endure an infinite number of cycles without failing. This concept became crucial in design practices, as it allowed engineers to determine safe operating stress levels for certain materials. The concept of the endurance limit also highlighted the importance of material selection and the need for materials with high fatigue strength in critical applications.

In the 1930s, the stress-life curve, also known as the S-N curve, emerged as a significant tool for characterising fatigue behaviour. Researchers such as G.M. Sines and A.P. Coffin established a relationship between the applied stress (S) and the number of cycles to failure (N) for a given material. The S-N curve provided a graphical representation of fatigue behaviour, allowing engineers to estimate the fatigue life of a component under different stress levels. This curve became a fundamental tool in fatigue analysis and design.

Further progress in understanding fatigue occurred in the 1950s, with a focus on crack initiation and propagation. Researchers such as A.A. Griffith and E. Orowan introduced fracture mechanics concepts to explain the role of small cracks in the fatigue process. They emphasised that fatigue failures were initiated by microscopic cracks and that crack growth played a significant role in determining the fatigue life of a material. This understanding led to the development of methods for predicting fatigue life based on crack growth rates and provided insights into the mechanisms behind fatigue crack propagation.

Throughout the 20th century, advances in materials science and engineering contributed to improving the fatigue resistance of materials and the design of fatigue-resistant structures. Various techniques were developed to enhance the fatigue strength of materials, including shot peening, surface coatings, and the use of improved alloys. These methods aimed to modify the surface properties and microstructure of materials to resist crack initiation and propagation.

Additionally, design practices evolved to incorporate factors that influence fatigue behaviour. Stress concentration, notches, and fillets were considered to mitigate the effects of localised stress concentrations, which are prone to crack initiation. Engineering standards and guidelines were established to ensure safe design practices, taking into account the potential for fatigue failure.

In recent years, the study of fatigue has been further enhanced by the advent of computational tools and advanced experimental techniques. Finite element analysis (FEA) and other numerical methods allow engineers to simulate and predict the fatigue behaviour of complex structures, thus enabling the decrease of the safety factors and so of the costs. Advanced materials, such as composites, are also being extensively studied to understand their fatigue properties and develop new design approaches.

Today, the understanding of fatigue of materials continues to evolve. Researchers are investigating new materials, developing advanced modelling techniques, and exploring innovative ways to improve the fatigue performance of structures. The aim is to ensure the safety and reliability of engineering components subjected to cyclic loading conditions, thereby mitigating the risk of fatigue failure.

In summary, the history of fatigue of materials is characterised by a progressive understanding of the underlying mechanisms and failure modes associated with cyclic loading. It has led to the development of tools, techniques, and design practices that enable engineers to predict and manage fatigue failure, ensuring the reliability and longevity of structures and components.

2. FATIGUE THEORIES OVERVIEW

As it was introduced, this project uses the Theory of Critical Distances developed by David Taylor and it is presented in chapter 3. However, to understand its body and purpose, it is necessary to have a general knowledge in the field of failure and fatigue of materials.

Assuming a basic understanding of the mechanical properties of materials, this chapter serves as a brief review of the basic background in the physics of materials, introducing symbols and terminology related to the topic of this thesis. The primary focus lies on the deformation and failure of materials under stress, with specific emphasis on the explanation of the main basic methodologies of this area: stress-strain curve, stress-life curve, and fracture mechanics, as well as the importance of computer-based methods in this field, especially the finite elements analysis.

2.1. Stress-Strain Curves

The stress-strain curve, basis of the strain-life method, is one of the fundamental graphics to obtain or represent information about a material's behaviour in the fields of fracture and fatigue. Each material has an associated stress-strain curve which plots the stress as a function of the strain as its name suggests. However, to actually understand the information of this curve, it is necessary to understand how it is obtained, as this curve is the outcome of the tensile test, one of the most important tests to gain information of a material's mechanical response.

In order to perform the tensile test, a rod is gripped at one end and subjected to a controlled load at the other end, while the increase of the length is captured by an electronic sensor attached to the specimen. With the information of the load (P) and increase of the length throughout the process, it is possible to represent the engineering stress (S) as a function of the engineering strain (e) and thus to plot the stress-strain curve, where A_0 the initial area of the rod.

$$S = \frac{P}{A_0} \tag{1}$$

$$e = \frac{\Delta l}{l_0} \tag{2}$$

This engineering parameters are defined, as they can be measured during the tests. However, they provide instant values, therefore the continuous values must be defined, taking into account the changes in length and area derived from the test itself: the real stress (σ_{real}) and the real strain (ε_{real}). Here, it is the actual area (A), and not the initial one, the value to calculate the stress, A decreases during the test, increasing the real stress.

$$\sigma_{real} = \frac{P}{A} \tag{3}$$

$$\varepsilon_{real} = \int_{l_0}^{l} \frac{dl}{l} = \ln\left(\frac{l}{l_0}\right) \tag{4}$$

Which brings to the relation among the engineering and the real parameters:

$$\sigma_{real} = S(1 + e) \tag{5}$$

$$\varepsilon_{real} = \ln(1 + e) = \ln\left(\frac{A_0}{A}\right) \tag{6}$$

The first region of the function shows the elastic behaviour of the material, as many of them approximately obey the Hooke's law, i.e., stress is proportional to strain as it can be appreciated on Figure 1. The modulus of elasticity, also known as Young's modulus (E), is the constant of proportionality of this region, enabling the use of Hooke's law to define the curve in the early stage as:

$$S = Ee \tag{7}$$

The elastic or proportional limit denotes the stress value for which the elastic behaviour ends and, for most of the cases in engineering, the plastic one begins. In this phase the specimen suffers a rearrangement of its molecular structure which leads to new positions of equilibrium, such as dislocation motion in crystalline materials. Although there are materials that lack this mobility, the so-known brittle materials, in engineering ductile ones are the most common, as metals usually belong to this classification. They have a plastic behaviour before failure, which conforms to a straight line on logarithmic axes of real stress against real plastic deformation and can be expressed as:

$$\sigma_{real} = K\varepsilon_{real,plastic}^{n} \tag{8}$$

Where K is the strength coefficient and n is the strain hardening exponent (0 for fully plastic solids). Considering a total strain equal to the sum of the elastic and plastic strain, it is possible to obtain a real fracture value at which the curve ends.

$$\varepsilon_{real,total} = \frac{\sigma_{real}}{E} + \left(\frac{\sigma_{real}}{K}\right)^{\frac{1}{n}} \tag{9}$$

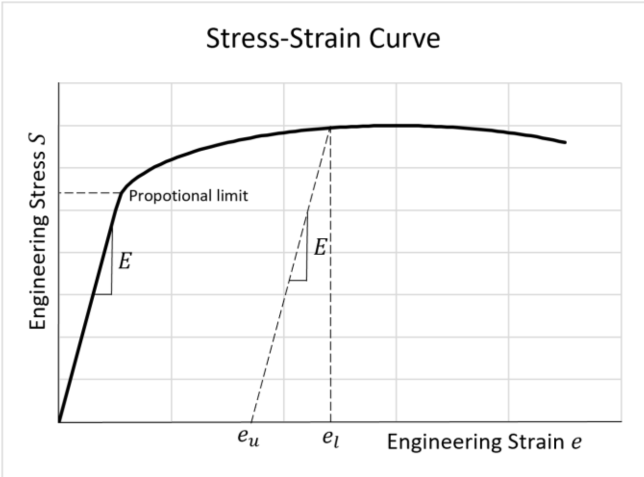


Figure 1: Example of a stress-strain curve obtained with the tensile test.

Plastic deformation refers to the deformation that cannot be reversed unlike the elastic one. If a specimen is subjected to a stress lower than the elastic limit, it comes back to its original measurements and form when the load is no longer applied, not occurring for plastic deformation, which remains without the need of any stress. Nevertheless, it is important to note that a workpiece under a stress higher than the elastic limit suffers a plastic deformation which remains but also an elastic deformation which reverses after the application of the load. To understand this phenomenon, it is better to use an example. On Figure 1 there is a point on the plastic region of the curve with a strain of e_l while the rod is loaded, nevertheless, this value does not remain when unloaded, but strain is then e_u, which can be obtained by drawing a line with slope (E) from the point on the curve in question to the x-axis, as shown in the example.

Another characteristic to take into account is that the material needs an increase of stress to continue the deformation, this phenomenon is called strain hardening.

One of the most important outputs of this curve is the yield stress (σ_y), being the needed stress to induce plastic deformation. However, it is often difficult and unclear when the plastic behaviour starts and thus the value of this material property, therefore it is typically defined as the needed stress to induce a residual plastic strain of 0,2%.

Another basic material property that can be reached with the information of this curve is the ultimate tensile strength (σ_{UTS}). Figure 2 where it can be found, at the pick of the curve, as it refers to the highest stress the material can withstand. Beyond that point the rate of stress hardening decreases and so does the required stress to increment the strain. This is actually not happening in reality but is a consequence of the use of the engineering parameters, since the decrease of the area, or "necking" is not considered with this approach, and it is after the σ_{UTS} when this phenomenon significantly rises.

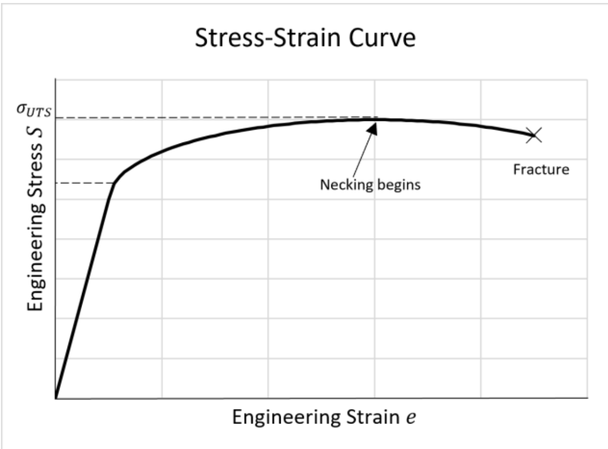


Figure 2: Ultimate tensile strength on the stress-strain curve

The stress- strain curve is a fundamental tool used to design and study the behaviour of materials, not only for avoiding failure, but also for the design of mechanical components.

2.2. S-N Method

The stress-life or S-N method is one of the bases for the study of fatigue of materials, as it was the first method developed in this field more than 100 years ago and it is still used for some design solutions. In this section, an introduction to it will be presented, but first it is convenient to define some of the main features of the fatigue of materials.

As stated in the introduction, fatigue of materials focuses on the study of the behaviour of bodies subjected to cyclic loadings, either constant or variable in time. For a simpler explanation and understanding, variable amplitude cyclic loads are left apart in this project, as they do not play a role in it and are more advanced, nonetheless, they are present in the majority of the cases in real life and are worth being aware of.

In order to continue with the explanation, a sinusoidal wave, like the one of Figure 3, will be used as the example load due to its simplicity. The first step would be defining the characteristic parameters of the wave, which are: the maximum stress (σ_{max}), the minimum stress (σ_{min}), the mean stress (σ_{mean}), the stress amplitude (σ_a), the stress range ($\Delta\sigma$) and the stress ratio (R):

$$\sigma_a = \frac{\sigma_{max} - \sigma_{min}}{2} = \frac{\sigma_{max}}{2}(1 - R) \tag{10}$$

$$\sigma_{mean} = \frac{\sigma_{max} + \sigma_{min}}{2} = \frac{\sigma_{max}}{2}(1 + R) \tag{11}$$

$$R = \frac{\sigma_{max}}{\sigma_{min}} \tag{12}$$

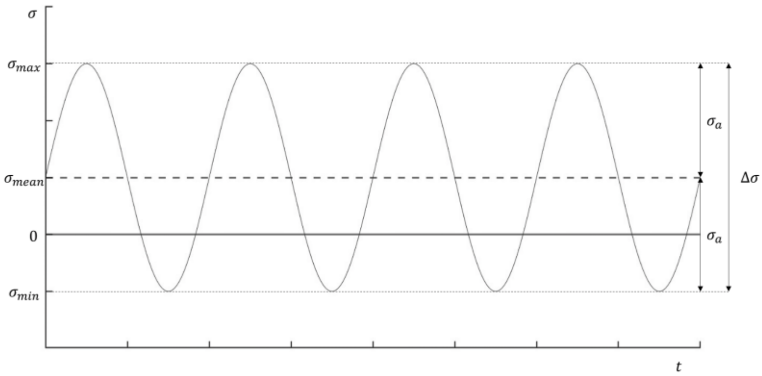


Figure 3: Stress variables for constant amplitude cyclic stress

2.2.1. S-N Curves

The fundamental element on which the S-N method is based is the S-N diagram or curve, developed by Wöhler, which graphically relates the stress amplitude, σ_a, to the number of cycles N_f to failure (Figure 4). This curve is obtained empirically by interpolation of the points found for each test and is plotted on a logarithmic scale due to its linear tendency in this form (Figure 5) and it can be mathematically described between 10^3 and 10^6 with the "Basquin equation" (13) where σ_f' is the fatigue strength coefficient and b is the Basquin exponent[1].

$$\sigma_a = \sigma_f'\left(2N_f\right)^b \quad (13)$$

[1] See "Y.B. Liu, .Y.D. Li, S.X. Li, Z.G. Yang, S.M. Chen, W.J. Hui, Y.Q. Weng (2010) *Prediction of the S–N curves of high-strength steels in the very high cycle fatigue regime*" for more information.

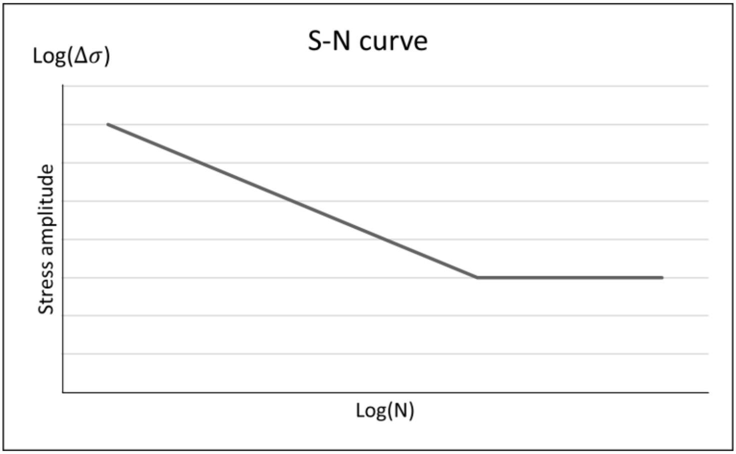


Figure 4: Theoretical S-N curve

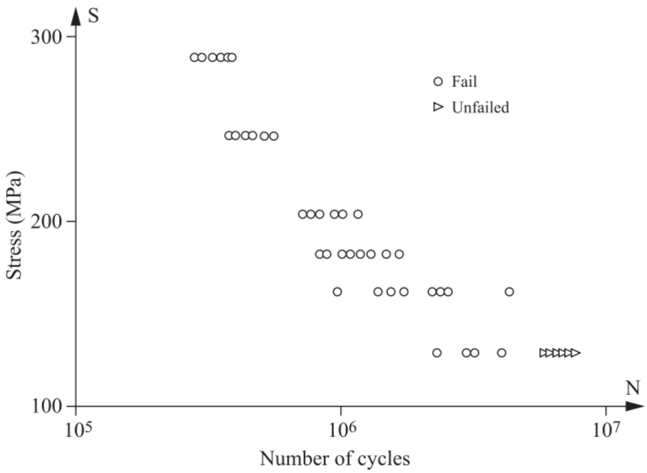


Figure 2.1. *Presentation of the test results using a Wöhler diagram (S-log N)*

Figure 5: Presentation of the test results using a Wöhler diagram (S-log N) [2]

Two main regions can be defined in this graphic known as low cyclic fatigue (LCF) and high cyclic fatigue (HCF). The first one refers to the region where the component experiences plastic deformation for each cycle and therefore, failure occurs at a lower number of cycles. While the HCF includes the cases for which the deformation is only elastic for each cycle, needing a higher number of cycles to break.

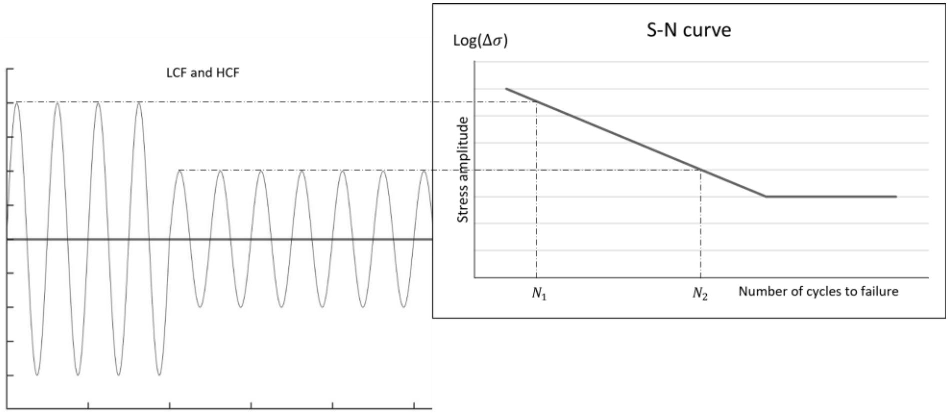


Figure 6: Example of a Wöhler diagram for high and cyclic fatigue

The main weakness of this approach is that it does not consider the real behaviour between stress and deformation and considers all the deformations as if they were elastic. For this reason, this method is usually employed for cases of long-life metals such as Body-centred cubic (BCC) Steel where a fatigue limit (σ_{FL}) can be found. The fatigue limit represents a value for which the material is considered to have infinite life and can be defined in these cases for the value of 10^6N, since from this point the curve becomes a horizontal line as in Figure 4. However, in many cases, a clear asymptote is not observed, and the fatigue limit is defined at a specific number of cycles, often referred to as the fatigue strength.

Recent research has revealed that the fatigue limit can sometimes be misleading. In some materials, failures can occur at very high numbers of cycles, exceeding 10^9N cycles, even at low stress ranges. Besides that, the alteration of any parameter has to be considered, for example the variation of the mean stress or the load ratio (R) shifts the entire stress-life curve (Figure 7).

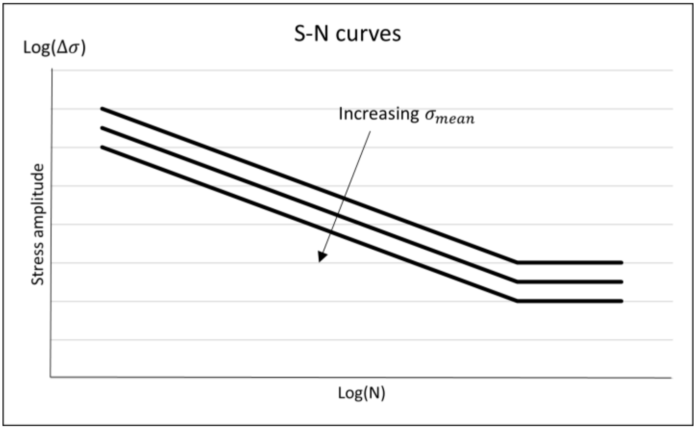


Figure 7: Variation of the S-N curve due to the mean stress

In this section the fundamentals of the S-N method have been introduced due to the importance of its understanding, as it is the basis of the fatigue study. However, there are many more factors that affect the fatigue strength of materials, such as size, shape, type of load, environment, etc. For this reason, nowadays, more advanced calculation methods are generally used, which consider more parameters and, consequently, are more realistic.

2.3. Fracture Mechanics

Fracture mechanics is a highly significant field of study that focuses on the behaviour of materials containing cracks. It has provided valuable insights by demonstrating that, under specific conditions, crack propagation can be predicted using simple linear elastic analysis. This branch of fracture mechanics is known as Linear Elastic Fracture Mechanics (LEFM).

Failure may occur due to multiple reasons, such as environment high loadings, design inaccuracies, defects of the material, production errors, etc. This makes the design against fracture a wide area of studies in technology where research is constantly carried out with the goal of approximating the calculations to the reality as accurately as possible. An introduction to this field is given in this section to show a basic understanding of it. Nevertheless, for designing purposes, it is convenient to consider as many factors as possible.

In this project the focus lies on notched specimens, where stress concentrations are found around the notches, as well as it happens with cracks. This stress concentrations are usually the cause of failure; therefore, it is the focal point of this section, and it is addressed within the field of fracture mechanics. First an introduction to basic fracture mechanics and second, in the following sections, a deeper look into notched specimens is given.

Due to the difficulty of the nomenclature from now on, Table 1 intends to present a clarification of the terms used below for a better understanding of the explanations. It presents and define different parameters named with the letter K, where their similar ones for fatigue are right next to the monocyclic loading ones.

Although metals like steel are often manipulated in engineering to increase their strength resistance, this improvement is accompanied by an increase in their brittle response as well, which means that any existing crack will propagate faster with less plasticity until the failure, leaving a shorter period to notice and react to the upcoming failure. This has caused many unfortunate accidents and therefore the importance of its understanding.

In any actual situation of a workpiece under a loading, the stress has different values over the body, i.e., stress gradients arise due to multiple causes, such as geometry, cracks, supports, notches, irregularities, etc. Particularly, cracks in the material cause stress concentration around its tips, leading to failure at stresses lower than the material can theoretically withstand. Fracture mechanics refers to failure prevention mechanisms in which a crack is assumed to exist, and the behaviour of the component is predicted in relation to the length of the crack, the material's resistance to crack growth and the threshold stress for crack propagation.

Table 1: Definition of the terms named with K in fracture mechanics.

Monocyclic Loading	Fatigue
K Stress intensity factor: it quantifies the magnitude of stress at the tip of a crack. It depends on crack size, applied stress, and geometry.	ΔK Stress intensity factor range: it represents the difference in stress intensity factors (Kmax - Kmin) experienced by the material during the loading cycle. It denotes the range of stress intensity factors acting on the crack during cyclic loading.
K_c Fracture toughness: it is the measure of a material's resistance to crack propagation in the presence of a crack or notch. It quantifies the critical stress intensity factor at which rapid crack propagation occurs.	ΔK_{th} Crack growth threshold: it represents the threshold stress intensity factor range below which crack growth is significantly reduced or negligible under cyclic loading conditions.
K_t Stress concentration factor: usually related to cracks or sharp notches, it represents the stress concentration factor at the tip of that mention crack or notch for the giving stress field conditions. This is the mathematical definition; however, this value is mostly obtained from empirical data. $K_t = \frac{\sigma}{S}$	K_f Fatigue Stress Concentration Factor: particularly related to cracks and notches, this stress concentration factor is a term used in the calculation of fatigue, because, unlike K_t, K_f accounts the effects of material properties and crack growth. This is the mathematical definition; however, this value is mostly obtained from empirical data. $K_f = \frac{S_{FL}(liso)}{S_{FL}(entallado)}$

Cracks, as well as irregularities or notches, produce stress concentrations, which Inglis studied how to calculate around elliptical holes, however, applying the same logic to cracks brings a mathematical difficulty, as at the tip of a perfect sharp crack stresses approach infinity. Of course, this does not represent the real physics, since that would mean that the material would break even for extremely small loads, which is not the case. For this reason, Griffith developed a different approach based on the energy balance, which led him to the Griffith equation:

$$\sigma_f = \sqrt{\frac{2E\gamma}{\pi a}} \tag{14}$$

$$\sigma_f = \sqrt{\frac{EG_c}{\pi a}} \tag{15}$$

This equation enables the calculation of the stress at which there is just enough elastic energy stored in the body to drive crack propagation. The so-called brittle fracture strength (σ_f) depends only on the crack length a, the critical strain energy release rate G_c and the Young modulus E. These two last parameters are material properties, as $G_c = 2\gamma$ and γ represents the surface energy, therefore it is possible to combine them, define the material toughness K_c and rewrite the expression as:

$$\sigma_f = \frac{K_C}{\sqrt{\pi a}} \tag{16}$$

Where:

$$K_C = \sqrt{G_c E} \tag{17}$$

This equation can be generalised by adding a geometrical factor F, which considers the variations depending on the shape of the component.

$$\sigma_f = F\,\frac{K_C}{\sqrt{\pi a}} \tag{18}$$

Also, this same relation can be adapted to cases of fatigue by switching the static magnitudes per cyclic ones:

$$\Delta\sigma = F\frac{\Delta K}{\sqrt{\pi a}} \tag{19}$$

Another important concept is the crack growth rate, $da/_{dN}$, which has been found to be influenced by the stress intensity range, ΔK. By understanding this relationship, it is possible to predict the rate at which cracks propagate in a given material under specific loading conditions.

Being the crack growth rate expressed as:

$$\frac{da}{dN} = A(\Delta K)^n \tag{20}$$

Where A and n are material constants, which depend on the stress ratio (R). This equation is expressed graphicly as on Figure 8, where two asymptotes can be found. The one on the left a growth threshold, which happens to be ΔK_{th}, below which crack growth effectively ceases, whereas on the right a second asymptote draws an upper limit above which the growth drastically increases, and brittle fracture appears. Changing the mean K, or R ratio, shifts the curve as shown.

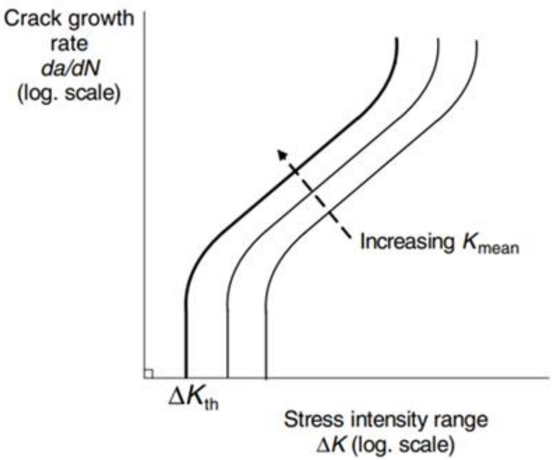


Figure 8: Typical fatigue crack growth rate curves [14]

It is worth noting that crack growth rate is influenced by various factors, including material properties, environmental conditions, and the presence of stress concentrations or corrosion. Experimental testing and analysis are typically employed to determine the crack growth rate and establish relationships between crack growth rate and stress intensity range for specific materials and loading conditions.

For the explanations presented so far, a body of constant thickness was assumed but no particular values for that thickness. However, on 3D case, as so is reality, depending on the thickness of the specimen, different stress conditions may arise.

For thin specimens, when the thickness is small, plane stress conditions occur, which means that the through-thickness stress is assumed to be zero. Whereas in thicker specimens, especially for metals and materials that develop plastic zones, the material near the crack tip in the centre of the specimen experiences plane-strain conditions, where it is finite and varies with the radial distance. This means that crack propagation is easier, and the fracture toughness is lower when plane strain is present.

2.4. The Failure of Notched Specimens

As mentioned in the previous section, the presence of notches leads to stress concentrations in their vicinity and thus to the appearance of stress gradients. Essentially, some notches behave similarly to plain specimens once the stress-concentration factor is accounted for. In these cases, failure occurs when the stress at the notch root reaches the strength of a plain specimen, whether under monotonic or cyclic loading. On the other end of the spectrum, certain notches exhibit behaviour akin to cracks of the same length. If the notch-root radius is small enough, failure is expected at $K = K_c$, or $\Delta K = \Delta K_{th}$ in cyclic loading.

However, the reality is that many notches do not follow these extreme cases. At failure, the stress at the notch root often surpasses the strength of a plain specimen, and the stress intensity factor can exceed K_c. Consequently, notches can be unexpectedly stronger, to the point where conventional calculations become inadequate, even as conservative estimates. Notches also demonstrate intricate size effects, influenced by both the notch size and the size of the specimen containing it. Thus, small notches (and cracks) can fail with a local stress greater than the plain specimen strength, but a stress intensity factor lower than K_c. Similar challenges arise when predicting fatigue failure.

To tackle these complexities, several methods have been devised. One commonly used in engineering design is the Neuber method, which characterises the behaviour using strain instead of stress. While this method proves useful, it has limitations, particularly at high stress-concentration factors and when attempting to predict size effects.

Ideally, a comprehensive theory that applies universally to stress concentrations is sought, encompassing sharp cracks, plain specimens, and stress-concentration features of non-standard shapes. One of theories that meet these requirements is the Theory of Critical Distances (TCD), further explained in chapter 3.

2.5. Finite Element Analysis

Over the past few decades, there has been a remarkable advancement in computing power, enabling engineers to employ sophisticated numerical analysis methods for simulating complex systems. This revolution has had a significant impact on engineering design. Nowadays, designers, even in smaller engineering companies, have access to techniques such as multi-body analysis and finite element analysis (FEA) to estimate forces and stresses in components. This shift has brought about a fundamental change in the design process, moving away from simplified analytical calculations and empirical rules toward computer simulations.

This numerical method is used in engineering and physics to solve complex physical problems, by dividing them into smaller, simpler, and manageable elements. These elements are interconnected at specific points called nodes, and FEA approximates the behaviour of the entire system by analysing these smaller components. For example, some equations or analytical methods do not have a solution for continuum systems, but they can be solved for discrete points, and that is the philosophy behind this method to become a continuum complex problem into many easier discrete problems to be solved and, by bringing this information together, creating a mesh of values that aims to represent the reality as accurately as possible. A finite elements analysis process often looks as it follows:

- **Problem Setup**: Identification and description of the physical system or structure to analyse. This could be anything from a mechanical component like a bridge or a machine part to a fluid flow problem or an electromagnetic field analysis.

- **Discretization**: Division of the complex system into smaller, finite elements. These elements can be simple shapes like triangles, quadrilaterals (for 2D problems) or tetrahedra, hexahedra (for 3D problems).

  Nodes are placed at the corners of these elements, and the variables of interest (e.g., stress, displacement, temperature, etc.) are approximated at these nodes.

- **Mathematical Formulation**: Statement of the governing physical equations for each element based on the behaviour of the material or system. Equations often derived from fundamental principles such as conservation of mass, momentum, energy, etc.

- **Assembly**: Combination of the equations for all the elements to create a system of equations that represents the entire structure or system. It typically includes stiffness matrices (for structural analysis) or mass matrices (for dynamic analysis) and load vectors.

- **Boundary Conditions:** Definition of the boundary conditions of the problem to address, such as fixed points, applied loads, constraints, etc.

- **Solution**: Solution of the system of equations, each node represents a small problem to solve, often using iterative numerical methods, to obtain the values of the variables of interest (e.g., displacements, stresses, displacement, etc.). These nodal values aim to satisfy both the governing equations and the boundary conditions.

- **Post-Processing**: Extraction of the needed information for the specific case of study, as it might be the generation of plots, graphs, and reports that illustrate the results, such as stress distributions or deformation patterns.

- **Validation and Interpretation**: Comparison of the FEA results to experimental data or analytical solutions if available or solutions for similar problems to validate the accuracy of the simulation. Interpretation of the results in the context of the problem in question in order to obtain the searched conclusions and improve the analysis if necessary.

It is important to note that the accuracy of any computer model is contingent upon the user's understanding of its boundary conditions, such as applied loads and restraints, as it requires expertise in both the numerical methods used and the physics of the problem being analysed. While FEA has some limitations regarding the size and complexity of the components that can be modelled, especially when considering non-linear and anisotropic material behaviour, the critical distance method implemented in this project primarily relies on linear-elastic stress analyses. The required stress-distance data can already be obtained for many engineering components using the standard FE models commonly employed in engineering.

2.6. Concluding Remarks

The goal of this chapter is to bring a summary of the fundaments of failure and fatigue of materials, which are essential for the comprehension of any further research in this field. Although it might appear a wide introduction, it only provides a basic and superficial information. In order to completely understand the physics behind these phenomena and be able to put these theories into practice (e.g. to design a component in engineering), each of these theories and methods should be studied in depth and numerous more factor should be considered for real life, like the nature of the environment for example, or the nature of the loads, which is never a perfect sinusoidal wave in real cases, as it is defined for the explanation of the theory.

The focal point of this project is theoretical, as it does not seek to design a workpiece for a real purpose but to state the potential of the TCD and Ansys. For this reason and for the sake of simplicity, this chapter provides this basic information and does not cover other specific but also important factors.

3. THE THEORY OF CRITICAL DISTANCES

3.1. Definition

The theory of the critical distances (TCD) is a group of methods, which can be gathered together in a theory due to their common features, being mainly a characteristic material length parameter, the critical distance L, which is used to carry out predictions of brittle fracture and fatigue in a simple way. Although this theory is only applicable to situations where the elastic stress field around the stress concentration feature is known, it has a high potential for numerous problems, since this information can easily be obtained by a Finite-Element-Analysis (FEA).

The TCD is composed of four methods incrementally ordered by difficulty of use: Point Method (PM), Line Method (LM), Area Method (AM) and Volume Method (VM).

3.2. Description and Applications

For the description of the TCD the two specimens of study in this project will be used as examples to make the explanation clearer. Both specimens are 2D planes with symmetrical notches; the first one, referred as "centre hole", has a circular hole in the middle of the plane (Figure 9), while the second one, the "V-notch", has notches which might resemble a V on each side of the plane (Figure 10).

As it has been stated previously[2], if a stress is applied to a component, its effect is not the same all over the area, but some parts experience higher stresses, these parts are called the stress concentration zones. This depends on the geometry, notches, small cracks, or irregularities on the material cause these stress concentrations, and

[2] See section 2.3 Fracture Mechanics.

therefore, it is there where the brittle or fatigue failure starts. For the two presented examples it is easy to see that the stress concentration zones are on the edges of the notches perpendicular to the applied load. For this reason, it is in the vicinity of those notches where the study must be more exhaustive since it is there where it would eventually break.

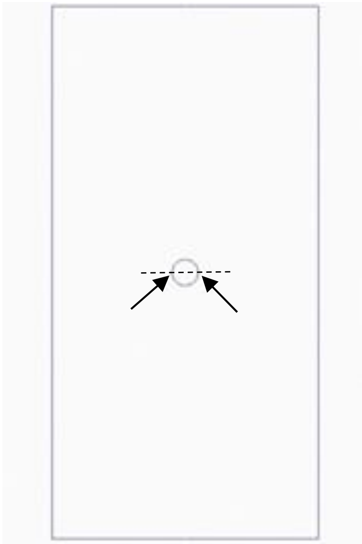


Figure 9: Centre hole plane

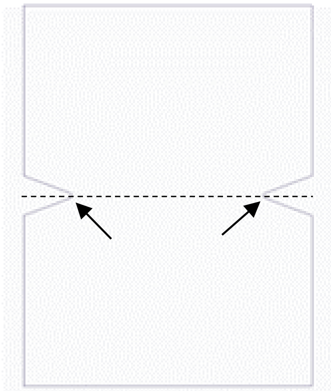


Figure 10: V-notch plane

In brittle fracture, considering a hypothetically perfectly symmetric and continuous specimen, as a 2D plane without notches could be, failure occurs when the applied tensile stress is equal to the ultimate tensile stress, since that is its definition as material property. In the case of having notches or irregularities, like in the presented cases, the fracture occurs under a lower tensile stress, due to the stress concentration. For these cases the TCD is used to calculate which would be the minimum value of applied stress necessary for the component to break.

The Point Method is the simplest of the methods belonging to this theory, as well as the one used for the later analysis of this project. Therefore, it will be the PM, the one explained and introduced first for the explanation of the TCD.

3.3. Point Method

The Point Method (PM) is the simplest approach of the TCD, this theory states that the specimen breaks when the stress in a specific point reaches the value of the critical stress σ_0. Hence, to carry out this method it is necessary to know:

- Two material parameters, L (Taylor's length) and σ_0 (critical stress), considered the minimum stress affecting the critical point for the specimen to break. These parameters are obtained using the material property values.

- The stress analysis of the region near the notch, which will be an elastic one, as it is assumed that no permanent strains or non-linear stress-strain behaviour occurs. This unrealistic assumption does not vary the results importantly; however, it highly simplifies the calculations. In the majority of cases, the calculation of the stress-distance curve is done using FEA or other numerical methods, as analytical solutions are only available for a reduced number of simple cases. In this specific project FEA is used throughout all the work.

With this data and a simple linear comparison, the maximum stress applied on the specimen before the break can be found. The "centre hole" and "V-notched" specimens are presented for the following example for a better understanding.

The location of that mentioned critical point, referred to as Q from here on, needs to be defined for each case. Figure 11 and Figure 12. show where Q can be found in both cases studied for this work and how it has been defined.

First the point of maximum stress (A) is found, which is not always easy to do but it is in this case, as the maximum stress will be reached in the edge of the notch. Second, the path of maximum stress can be defined as the straight line departing from A perpendicular to the tensile stress. Lastly, the Q point will be located at a distance of $L/2$ (half of the Taylor's length) from A on this path.

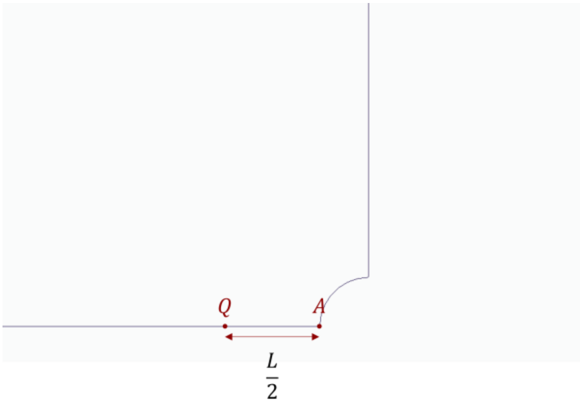


Figure 11: Representation of the PM on the centre hole plane

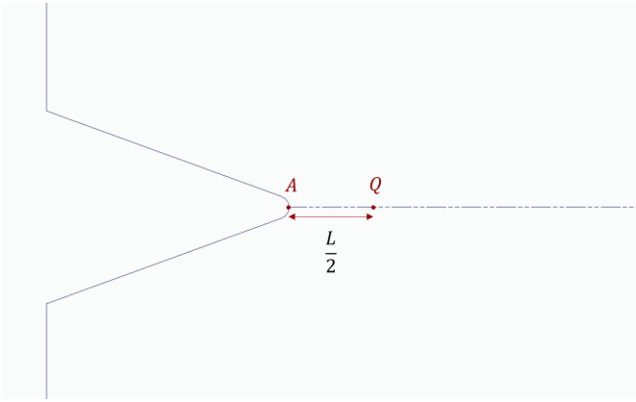


Figure 12: Representation of the PM on the V-notch plane

The TCD is a linear theory for the majority of cases, as it is for the two geometries that concern this study. This allows to calculate the maximum load that can be applied to the component with a relatively simple method, which can be presented in the following four steps:

- Run a FEA over the specimen with a basic applied stress ($S_{a,1}$), for example: 1 MPa, 10 MPa, 100 MPa, etc.
- Find the value of the (maximum principal) stress in the Q point, $S_{q,1}$
- Thanks to the linear behaviour, the maximum possible applied stress over the component before failure ($S_{a,0}$) can be defined as it follows:

$$S_{a,0} = S_{a,1}\left(\frac{\sigma_0}{S_{q,1}}\right) \tag{21}$$

- Run the FEA again but applying $S_{a,0}$ as load and confirm that $S_{q,1} = \sigma_0$.

The only remaining question would be how to calculate the critical stress σ_0 and the Taylor's length L. These two values can be deduced from empirical data, carrying out an experiment, in which two specimens are the object of this test. For the sick of simplicity, the two extreme conditions are studied to find the wanted values: a sharp crack and a plain (unnotched component).

In case of a plain specimen, there is no stress concentration or gradient, so failure occurs at a tensile stress equal to the ultimate tensile stress, giving the trivial solution of equation (22).

$$\sigma_0 = \sigma_u \tag{22}$$

On the other hand, for a specimen with a long, sharp crack, it is possible to link the TDC, in this case the PM, to the LEFM, introduced in section 2.3. Going back to what was explained previously then, failure occurs when $K = K_c$ and K_c is related to σ_f as:

$$\sigma_f = \frac{K_C}{\sqrt{\pi a}} \tag{23}$$

While the stress–distance curve starting from the tip of a crack can be expressed analytically as:

$$\sigma(r) = \sigma\sqrt{\frac{a}{2r}} \tag{24}$$

Combining both expressions for the case of $r = {}^{L}/_{2}$ and $\sigma_0 = \sigma_u$, L can be defined as:

$$L = \frac{1}{\pi}\left(\frac{K_c}{\sigma_u}\right)^2 \tag{25}$$

This explanation has been carried out for the case of fracture mechanics; however, the same procedure can be followed for fatigue, leading to the following analogous equations:

$$L = \frac{1}{\pi}\left(\frac{\Delta K_{th}}{\Delta\sigma_u}\right)^2 \tag{26}$$

$$\Delta\sigma_0 = \Delta\sigma_u \tag{27}$$

This is valid for a wide variety of cases, such as brittle fracture of ceramic materials, and fibre composites and fatigue of metals, but not for brittle fracture of polymers or metals.

For this project a widely used material was chosen, the stainless steel. Its properties were acquired from the following table:

Table 2. Variations on a critical distance theme. (Variations on a critical distance theme, 2021)

Materials and geometries of notched components from the literature.

Material	$\Delta\sigma_{FL}$ (MPa)	σ_{YS} (MPa)	ΔK_{th} (MPa)	L (mm)	L^* (mm)	Geometry[a]	Loading	R	Reference
0.46% C steel annealed	480	284	10.42	0.150[b]	0.257	CHB	Rot. Bending	−1	Murakami [26]
0.13% C steel	362	206	11.00	0.294[c]	0.521	CHB	Rot. Bending	−1	Murakami [26]
Al alloy 2017-T4	313.8[d]	368.7	11.51	0.428[i]	0.506	CHB	Rot. Bending	−1	Murakami [16]
Brass 70/30	245.2[d]	103	6.39	0.216[i]	0.522	CHB	Rot. Bending	−1	Murakami [16]
0.37% C steel	470	328[e]	15.36	0.340[f]	0.515	CHB	Rot. Bend., Axial	−1	Endo [27]
1045 steel	606	466	13.84	0.166	0.236	CHP	Axial	−1	DuQuesnay et al. [28]
Al alloy 2024-T351	248[g]	360	7.09	0.260	0.291	CHP	Axial	−1	DuQuesnay et al. [28]
Al 7075	516[h]	595	7.76	0.072[i]	0.086	CHB	Axial	−1	Chaves et al. [29]
Stainless steel	632	467	15.03	0.180[j]	0.262	CHB	Axial	−1	Chaves et al. [30]
Mild steel	400	293.4[k]	12.88	0.330	0.483	ENP	Axial	−1	Frost et al. [31,32]
Mild steel(2)	446	339.7[l]	12.99	0.270	0.386	CNB	Axial	−1	Frost et al. [31,32]
Al alloy B.S. L 65	300[m]	432.3[l]	4.19	0.062[c]	0.070	CNB	Axial	−1	Frost [33]
SM41B steel	326	194	12.36	0.458	0.781	CNP	Axial	−1	Tanaka et al. [34]
15313 steel	440	380	12.01	0.237	0.316	CNB	Axial	−1	Lukas et al. [35]
Al alloy AA356	231[n]	192	3.95	0.093	0.127	CNB	Rot. Bending	−1	Atzori et al. [36]

From this table the fatigue limit stress and the fatigue threshold were obtained, $\Delta\sigma_{FL} = 632MPa$ and $\Delta K_{th} = 15{,}03MPa\sqrt{m}$ [3]. Using the expression (26) it is easy to calculate the value of the critical distance for this material, which is $L \approx 0{,}18mm$, as the table shows.

[3] In the table used another unit is stated, but this is a typo. The right unit is $MPa\sqrt{m}$

4. POINT METHOD IN APDL PRODUCT LAUNCHER

The first part of the project was to seek the stress concentration coefficients depending on the radius of the notch and represent this relation as a function on a graph. The tool to find these theoretical values has been the TCD, more specifically the Point Method or PM.

As it has been explained, any method of the theory of the critical distances is based on the stresses over the specimen. This calculation is mainly driven by a finite elements' software, making the process easier, faster, and possible for many difficult geometries, which do not have an analytical solution. In this study, the Mechanical APDL Product Launcher of Ansys has been the used software for these numerical calculations. In order to better understand the process, the next point, 4.1., provides a short introduction on what is Ansys and, more precisely, Mechanical APDL Product Launcher or also called APDL tool.

4.1. Mechanical APDL Product Launcher

ANSYS is a powerful engineering simulation software used to design, simulate and analyse various physical phenomena in multiple engineering fields. It enables obtaining a comprehensive suite of simulation solutions to optimise designs, reduce development time and cost, and improve product performance and reliability. ANSYS uses numerical analysis techniques such as finite element analysis, computational fluid dynamics, and electromagnetic simulation to predict how a product or system will behave under different operating conditions.

Mechanical APDL Product Launcher is a utility tool provided by ANSYS, which is used to launch the Mechanical APDL (Ansys Parametric Design Language) program, as its name suggests. This product is a powerful tool for solving complex engineering problems related to structural analysis, thermal analysis, fluid dynamics, and other related fields.

The Product Launcher provides a coding interface to set up and launch the Mechanical APDL program with the required input files and parameters, the code. Making it possible to reuse, change or adapt simulations, since the features of the analysis, parameters, conditions, etc., can easily be changed in the code, i.e., it is not necessary to start over again for every simulation, just adapt a previous code by changing only the specific features. Although this might seem obvious for every software, it is not so simple or even possible for all of them.

Overall, the Mechanical APDL Product Launcher simplifies the process of launching and running simulations, it enables to easily change some features/restrictions or use a previous design as start for a new project, since this can be done by adapting the input code. However, it also requires the capacity and knowledge of the user to be able to write or work with a code in this language.

4.2. Point Method Implementation to APDL

In order to represent the relation between the stress concentration coefficients depending on the radius of the notch as a function on a graph, numerous tests were carried out with APDL for different radius values. These values and the stress concentration factors found from each test were saved in an excel sheet, where they were interpolated to represent the wanted plots (Figure 20 and Figure 21).

In the case of the specimen with a circular hole in the middle, $1/K_f$ is represented against the logarithm in the decimal base of the notch radius, while in the case of the plain with the V form notches, $1/K_f$ is represented against the stress concentration factor, K_t, which is equivalent to the maximum stress, i.e., the stress on the edge of the notch (A point in Figure 11 and Figure 12.). For this reason, these values were sought depending on the radius in order to later interpolate them.

As stated, APDL has been the tool used to run the FEA over the specimens, but not only that, the Point Method has also been implemented and run in the codes, thus directly giving K_f and K_t as output. Although a different simulation was run for each point of the function, two main codes were created, one for each geometry, and only some small features were changed for the different radii, such as the input of the radius value or some specifications in the construction of the mesh, but only small adaptations. Therefore, the first step of the project was writing these two basic codes and ensuring they were correct.

The code is composed of three main parts, the definition of the specimen and its geometry, the finite element analysis, and the output of the wanted values. For the geometry, symmetry was used twice in each workpiece, as both have two axes of symmetry, thereby reducing the number of calculations to be carried out by the compiler. Figure 13 and Figure 14 show the symmetry axes and the quarters chosen to be defined on the code.

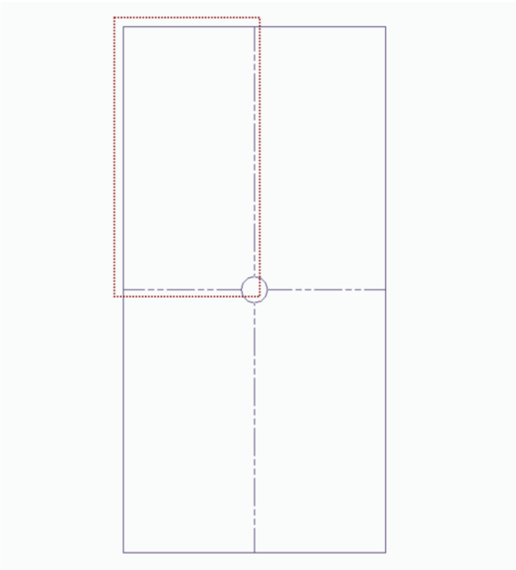


Figure 13: Symmetry of the centre hole plane

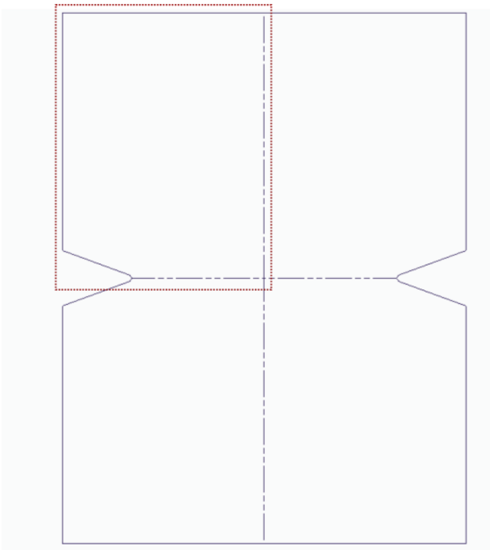


Figure 14: Symmetry of the V-notch plane

- Dimension: in this first section, all the dimensional input values are declared for their use throughout the rest of the code. It is worthy of comment that the stress applied over the specimen has a different value for every test, since what has to remain constant throughout the test is the effective stress on the horizontal middle line, the one of maximum stress, as it would break there and it is the focus of this theory. The width of this line changes for every radius value and so has to change the load applied on the upper line with the goal of always having 1 MPA in the middle line.

- Definition of the material and the specimen: input of Young's modulus = 200 GPa and definition of the type of geometry, in this case a 2D element of 8 nodes, where plane stress is assumed. In theory the test should be run over a perfect infinite plane, which is not realistic, but it can be simulated by a 2D plane ruled by plain stress, where the area of the specimen is large enough compared to the length on the notch, thus providing practically the same behaviour as in the case of an infinite plane.

- Key points of the geometry: definition of the needed points to build the geometry, which correspond to those illustrated on Figure 15 and Figure 16.

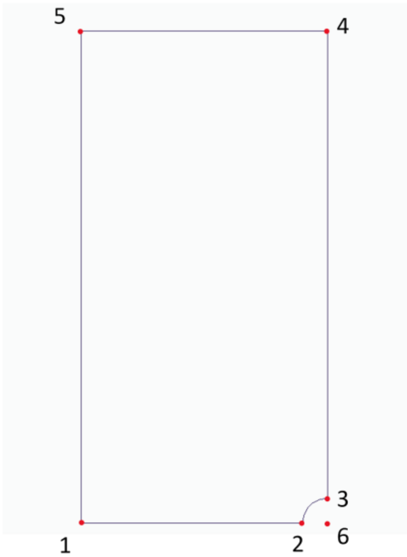


Figure 15: Key points of the centre hole plane

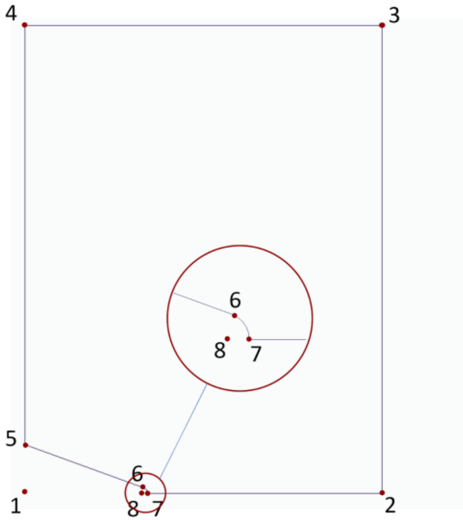


Figure 16: Key points of the V-notch plane

- Lines: in this part the lines shaping the geometry are declared (Figure 17 and Figure 18), as well as the area itself, hereby finishing the definition of the specimen and its geometry.

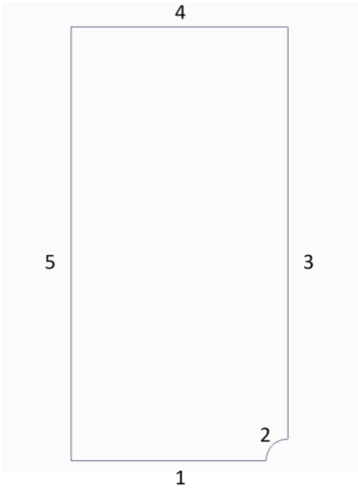


Figure 17: Lines of the centre hole plane

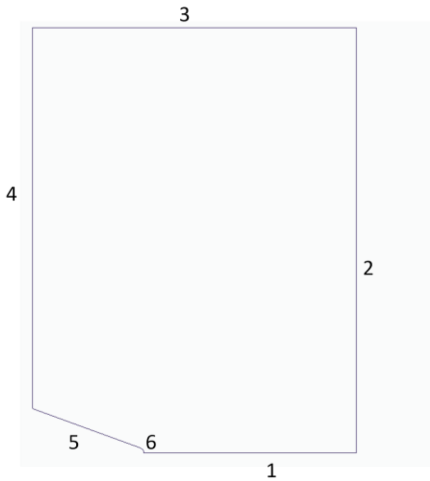


Figure 18: Lines of the V-notch plane

- Meshing features: in this section the mesh is created in order to carry out the FEA. In the zone close to the notch the calculations need to be more accurate, since it is the scope of this study. Therefore, the lines are divided in parts in a logical way, having smaller divisions in the vicinity of the notch and bigger ones in the upper zone, with the purpose of reducing the number of calculations run by the program. These divisions are used as the basis for the construction of the mesh, which can be refined around a point or a line, as it is shown in the code (see APPENDIX: APDL CODES). These refining functions are highly important in the process of confirming the accuracy of the results as it will be explained later in this chapter.

- Boundary Conditions: information about the symmetries in the workpiece and the applied tensile stress. With this information, the FEA can be solved.

- Representations: plot of stresses and displacements over the mesh to ensure that the solution is correct in the first instance.

- Path with the stress for the Point Method: definition of a path across the line 1 with starting point on the edge of the notch. This path consists of one hundred points and has a length of L, the critical length of Taylor.

- Getting scalar values: as end of the code, the two main outcomes for this project are obtained, the maximum stress and the stress on the point Q of Figure 11 and Figure 12., at a length of $L/2$ from the edge of the notch. This stress on Q is already the value of K_f, as the effective stress is 1MPa.

4.3. Procedure of Calculations

Once the draft codes for both geometries were built, the calculations for each point were carried out. Two main questions arise at this stage: How were the points chosen, in other words, the values of the different radii, and how was determined the accuracy of the results?

- Radius values: this was based on common sense. The plane should still behave as an infinite one, which means, the length of the notch had to be small enough, besides that, it was already known that these plots should approximately look like other ones based on empirical results and previous studies. Therefore, various ranges of radius values were tried out and numerous points within those. After the run calculation for many radii, around twenty points were selected for the interpolation and so the creation of the functions. Hence, during the calculation process, every third or fourth point or so, the function was plotted, to ascertain that these points were on the proper path and to select the next values of radii, seeking them in areas where there was a lack of accuracy.

- Accuracy of results: besides displaying the results on the graph, it was previously essential to verify the accuracy of the FEA and thus the outputs. In order to achieve this, the simulation of each radius was run more times applying some refining (KREFINE and LREFINE commands in the code) around the hot spot (edge of the notch where the stress is maximum) gradually increasing the density of the mesh on each simulation. The results were compared with the ones of one step before until the variation was lower than 1%. These regressions were carried out on an excel sheet for every radius value until the error was small enough, which took between 3 and 4 tries on average. Figure 19 shows an example of this procedure for the V-notch plane:
    - Column B shows the meshing part of the code.
    - Columns C and E respectively represent the maximum stress or K_t and the stress at $L/_2$ or K_f.
    - Columns D and F stand for the variation of the obtained results compared with the previous ones. This variation is determined with the following operation:

$$\%\Delta S_{max,n} = 100 * \frac{S_{max,n} - S_{max,n-1}}{S_{max,n}} \tag{28}$$

where $S_{max,n} \equiv$ Maximum stress of iteration number n.

Analogously, the same applies to the stress at $L/_2$ or K_f.

Iteration	Meshing features	S_max/Kt	%Variation	S_taylor/Kf	%Variation
1	LESIZE,1,,,200,100 LESIZE,2,,,20 LESIZE,3,,,20 LESIZE,4,,,100,0.01 LESIZE,5,,,100 LESIZE,6,,,50 AMESH,ALL ! Meshing	15.6697634		6.49420925	
2	! Meshing features LESIZE,1,,,200,100 LESIZE,2,,,20 LESIZE,3,,,20 LESIZE,4,,,100,0.01 LESIZE,5,,,100 LESIZE,6,,,50 AMESH,ALL ! Meshing KREFINE,7,,,, LREFINE,6,,,,	16.8395955	6.946913303	6.5354833	0.631537839
3	! Meshing features LESIZE,1,,,200,100 LESIZE,2,,,20 LESIZE,3,,,20 LESIZE,4,,,100,0.01 LESIZE,5,,,100 LESIZE,6,,,50 AMESH,ALL ! Meshing KREFINE,7,,,, KREFINE,7,,,, LREFINE,6,,,,	16.8628507	0.137907881	6.52606078	-0.144382964

Figure 19: Calculation of the accuracy of the results

With the collected data throughout this process, the points of both functions were interpolated as it was introduced at the beginning of section 4.2, obtaining the following graphs, which approximate highly accurately to reference graphs based on other empirical tests or theories.

Figure 20 illustrates how the function has two horizontal asymptotes, one of them at $1/K_f = 1$ and another one at $1/K_f = 1/3$. These data were to be expected, since the smaller the radius of the notch, the closer it is to a perfect plane without notches and therefore without stress concentrations. While as the radius increases, L does not change and is smaller and smaller compared to the notch, therefore the value of K_f gets closer and closer to the value of K_t, which in the case of a circle is always equal to 3.

On the other hand, Figure 21 shows how for small radius values (left part of the graph) the K_f increases rapidly with respect to the increase of K_t, however, from $K_t = 5$ onwards this tendency softens until it stabilises around $K_t = 10$, so that the K_f not only remains practically constant but also slightly decreases.

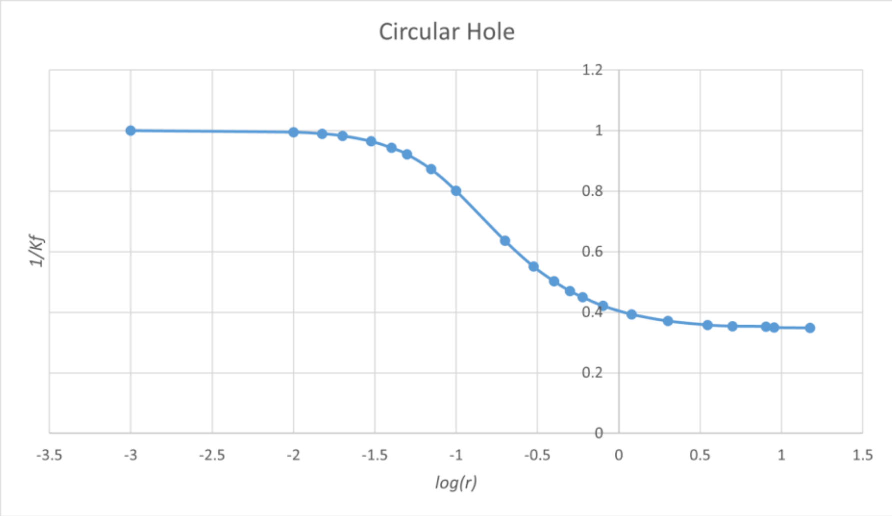


Figure 20: Variation of K_f depending on the radius.

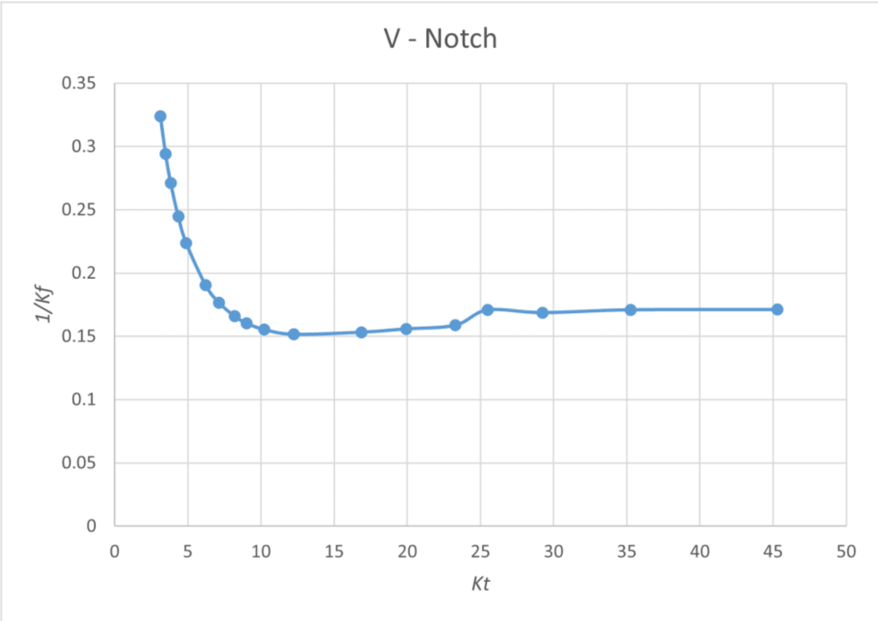


Figure 21: Relation between K_f and K_t depending on the radius.

5. IMPLEMENTATION OF THE FATIGUE TOOL OF WORKBENCH

Workbench ANSYS is a software platform developed by ANSYS., which provides a suite of tools for simulating and analysing engineering problems. Workbench integrates various ANSYS tools into a single graphical user interface, offering this way a more visual and simple approach to some complex problems like FEA. Due to the structure of this platform, the user does not have to be able to write a code or have a deep knowledge in the mechanical problem to be able to use the software.

ANSYS Workbench includes several tools and modules that enable different engineering analysis and simulation needs, such as Ansys Mechanical, Ansys Fluent, Ansys CFX, etc. However, the focus of this project is the study of fatigue of materials in the vicinity of notches, and how to predict their influence over the geometry with Workbench and APDL. Therefore, this chapter deeps into the understanding and good use of the Fatigue Tool, which can be found in the mechanical side of Workbench.

As it was shown during this thesis, the fatigue of materials is not a theory, but a relatively new area of physics and engineering. There is no definitive theory or analytical method that always provides the predictions of the fatigue damage. Instead, a pool of theories can be found, where each of them provide approximations to this damage in a different way, where the use of the one, another or various overlapped theories depends on the geometry, material, working conditions, environment, etc. The election of which of the theories to use and their development is not a simple job, it needs a high level of knowledge and experience in the matter. This brought the question, if Workbench has an automatic fatigue tool, how does it work and how accurate is it?

The first step was to go through the user's guide, to understand what the fatigue tool offers for the area of this study, fatigue in the vicinity of notches. However, some of the explanations in this guide are slightly unclear, therefore, an example was used to run simulations in order to compare the results with the analytical ones and achieve a better understanding on how the program works. The geometry used for this simulation was a plane with a hole in the centre, the same specimen, which was used as an object of study in APDL in the previous section.

The fatigue tool has to be applied after running a normal FEA over the specimen, which means before using this tool it is necessary to define the specimen, the boundary conditions, the load and run the finite elements analysis. For this reason, it is important to understand how to create a new project on Workbench. In the following section the procedure for this project is explained.

5.1. New Project Creation on Workbench

The first step is to open Workbench and click on file > new. The page appears empty then, as a type of project must be selected, for this case the "static structural", after choosing it the screen looks like on Figure 22.

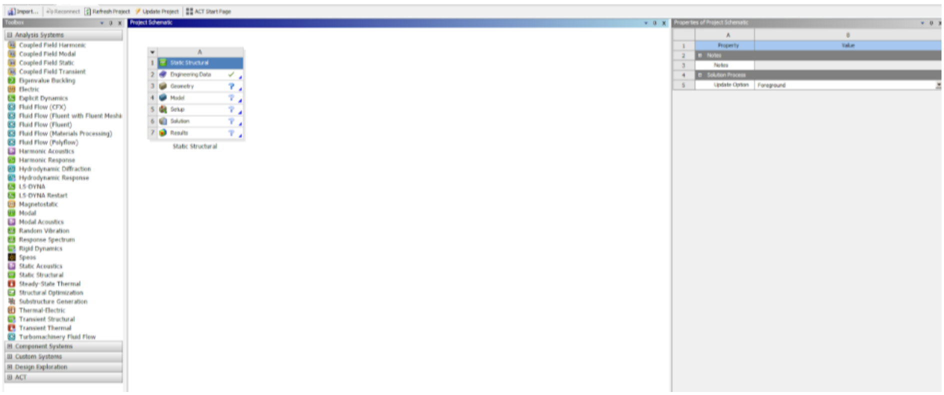


Figure 22: Workbench Static Structure

The second step is to define the material and its properties, which is done on engineering data. There are multiple possible inputs which can be given to the program, depending on the wanted analysis, different values are needed. For this project the same stainless steel was used as the one for APDL. The properties were obtained from Table 2; Figure 23 shows the S-N curve of this stainless steel and these values were also given to Workbench, since it is necessary for the later use of the fatigue tool. On Figure 24 the inputs of this project can be seen and how they were added to the program, the tensile yield strength, the compressive yield strength and the ultimate tensile strength are just values, but for the S-N curve, various points of the curve have to be given for the program to interpolate them as it can be seen on the right side of the Figure 24.

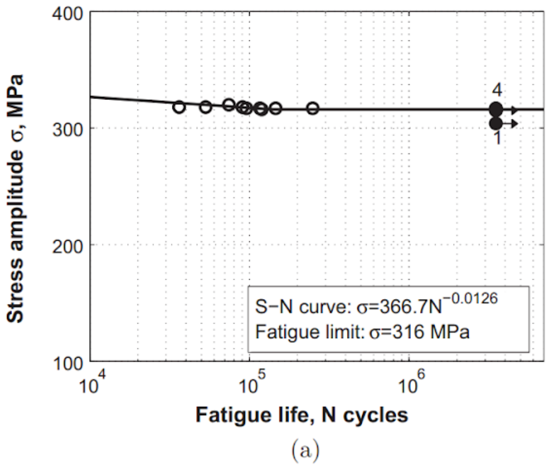


Figure 23: S-N curve of the used stainless steel [3]

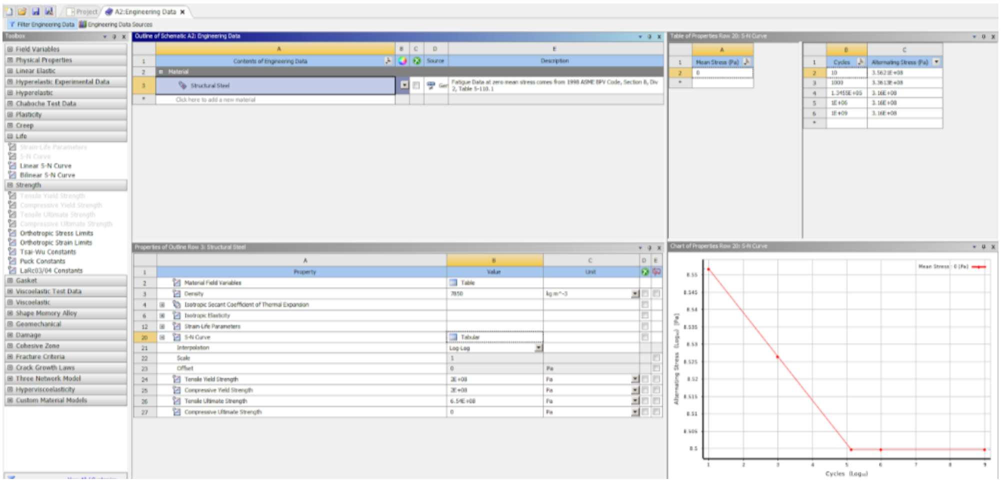


Figure 24: Workbench Engineering Data

After the engineering data the workpiece needs to be imported from another design software, such as Solid Edge or CATIA V5, or it can also be built with the design tool of Workbench, which opens by clicking on geometry. For this study, the geometry was designed with the New DesignModeler Geometry (Figure 25) and the analysis type had to be changed to 2D, as 3D is the default.

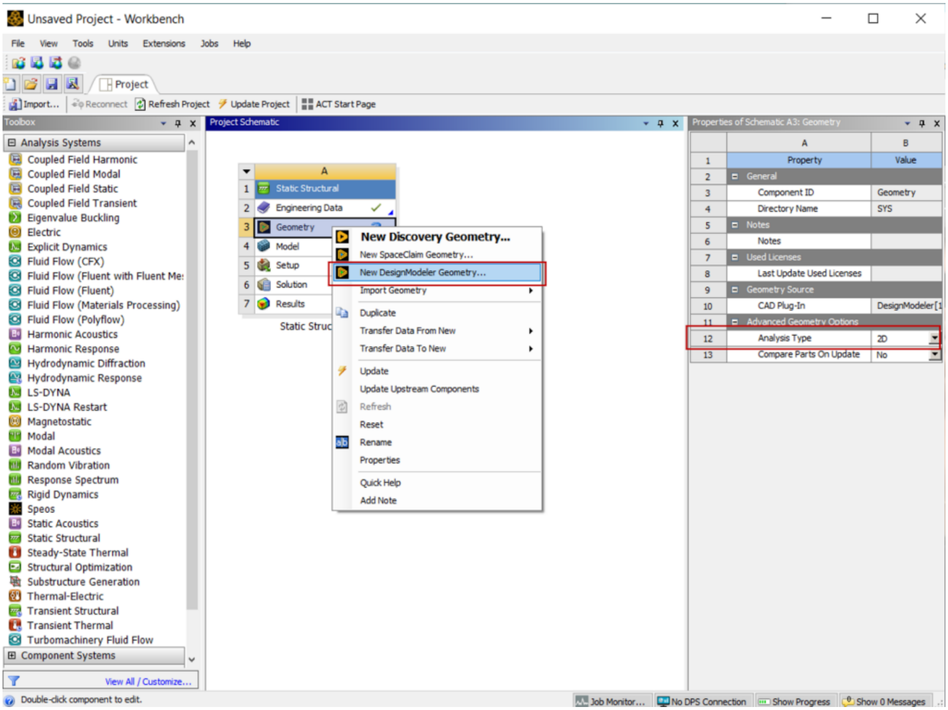


Figure 25: Workbench New DesignModeler Geometry

With the specimen finally defined, the work on the model can start and thus the finite element analysis. For this, it is convenient to have a background on how to run such an analysis, since the software automatically creates a mesh over the object, but its quality is debatable, hereby, the results are more accurate if the user adapts the mesh properly. In the example the area close to the edge of the notch was refined (Figure 27).

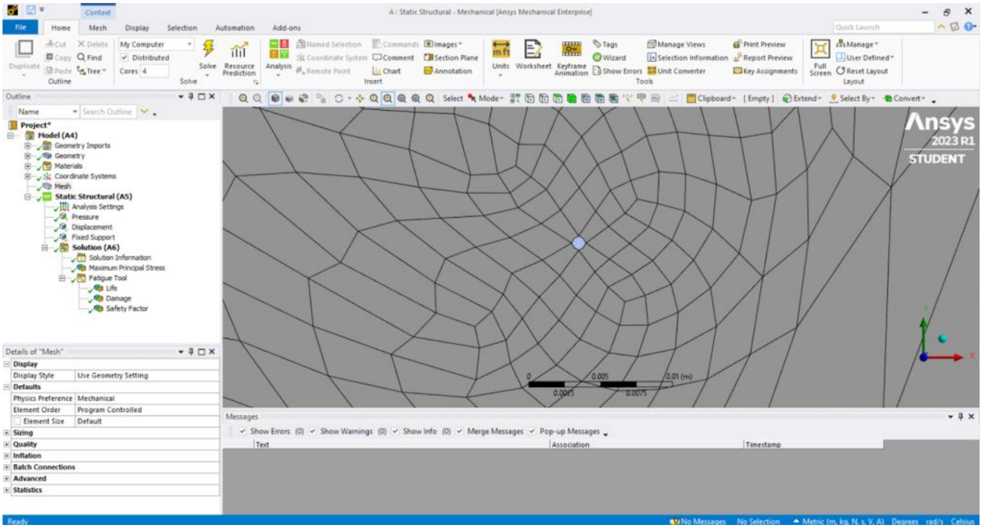


Figure 26: Automatic mesh

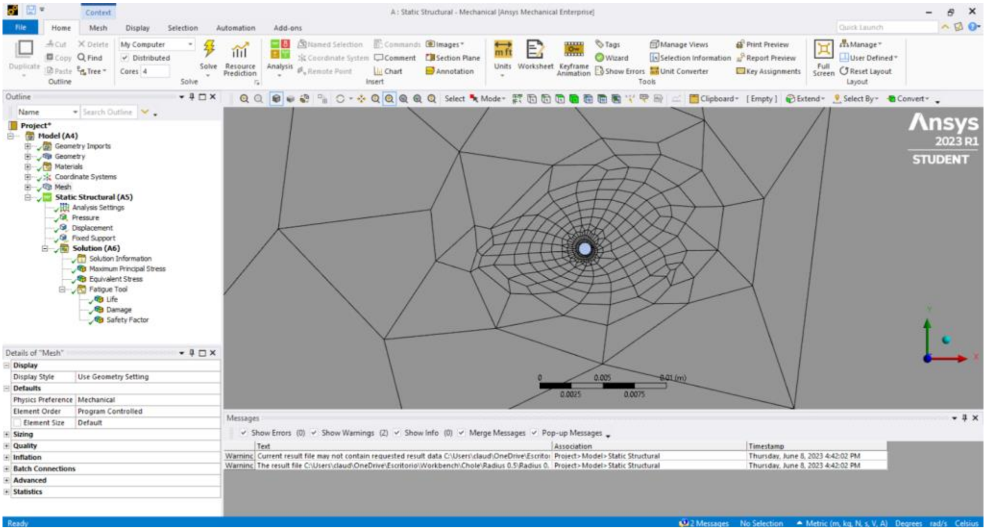


Figure 27: Mesh after the refine.

After adjusting the mesh in the areas of interest, it is important to define the right supports for the simulation. In the case of APDL this was not necessary, due to the use of symmetries, which stated the problem as a theoretical one without any physical support but with a certain tension stress on the edges. Although there is a symmetry function in Workbench, the fatigue tool cannot be applied to a geometry with symmetry in this interface. Thereby, a support had to be given, which is a delicate task, as these physical supports generate stress concentrations in the simulation, misleading the hot spot from the edge of the notch to the location of the support, which is not the matter of study in the current case. The finally chosen support for the simulation to avoid these unintended stress concentrations was a fixed support on the left lower corner of the plane and the restriction of displacements in the Y direction over the lower edge, allowing them in the x direction. The applied load was a tensile stress over the upper edge. (Figure 28)

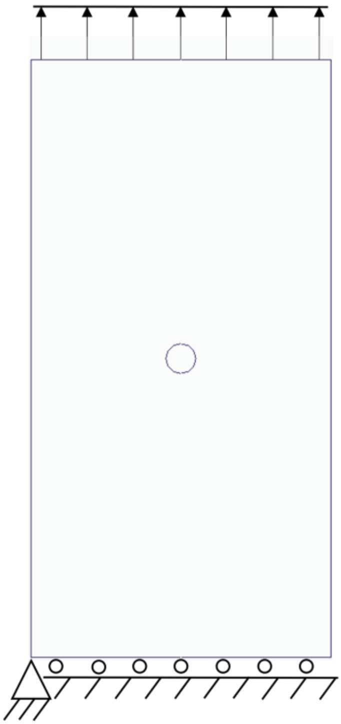


Figure 28: Representation of the problem in Workbench

Finally, the FEA can be run and many different results can be asked for, such as deformation, strain, stress or even the fatigue tool. For this project the outputs of higher importance are the maximal principal stress, the von Mises equivalent stress and, of course, the fatigue tool.

5.2. Fatigue Tool

The fatigue tool provides life, damage, and safety factor information and uses a stress-life or strain-life approach, with several options for handling mean stress and specifying loading conditions. This tool enables to perform frequency-based fatigue analysis using either the random vibration analysis, the harmonic response analysis, or a combination of both, which is highly tedious to conduct analytically or even impossible. The numerical methods of this interface facilitate these advanced calculations, but the difficulty of these problems lies on the number of calculations to do, and not on the complexity of the election and performance on a fatigue theory, which is not done by this tool, answering the raised question: How does the fatigue tool chooses a theory? It just does not. The user has to choose the approach, and there are only two of them as mentioned, the stress-life or strain-life approach.

A simple problem was chosen to understand better this tool is and its potential. For this, it is important to know what the inputs and outputs are, as they give an overview of what the program is capable of and what must be done by the user.

Inputs:

As it was already stated, this tool uses either a strain-life or a stress-life approach, having some different inputs depending on the approach. For the first one, the strain-life approach, four strain-life parameter properties and two cyclic stress-strain parameters must be given: the strength coefficient, the strength exponent, the ductility coefficient, the ductility exponent, the cyclic strength coefficient, and the cyclic strain hardening exponent.

On the other hand, the stress-life approach requires the definition of the stress-life or S-N curve, which can be defined in terms of mean stress, r-ratio, or temperature. The Interpolation method (Log-Log, Semi-Log, or Linear) can also be defined and, of course, the curve data must be greater than zero.

<u>Mean Stress:</u> This definition applies when experimental S-N data is gathered under

constant mean stress for individual S-N curves.

R-Ratio: This definition applies when multiple S-N curves are collected at a constant R-ratio.

Temperature: This definition applies when multiple S-N curves are collected at different temperatures. If the Temperature Field Variable is selected when defining a S-N Curve material property and multiple S-N Curves are given as input for different temperatures, the appropriate S-N curve is chosen for interpolation based on the temperature at each node of the body.

The S-N curve can be defined with its formula, linear or bilinear, or giving points of the curve in tabular (Figure 29, Figure 30 and Figure 31).

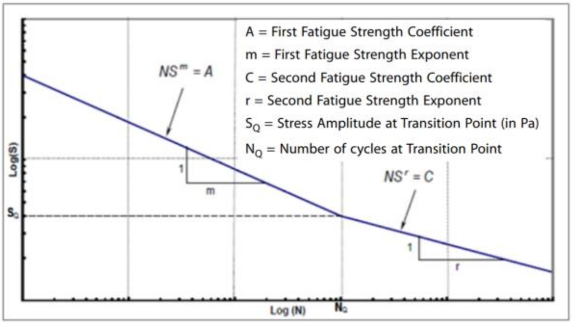


Figure 29. Bilinear S-N curve

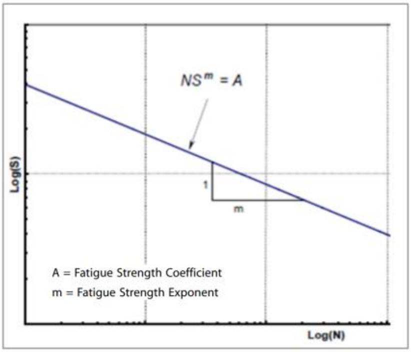


Figure 30. Linear S-N curve

	B	C
1	Cycles	Alternating Stress (Pa)
2	10	3.999E+09
3	20	2.827E+09
4	50	1.896E+09
5	100	1.413E+09
6	200	1.069E+09
7	2000	4.41E+08
8	10000	2.62E+08
9	20000	2.14E+08
10	1E+05	1.38E+08
11	2E+05	1.14E+08
12	1E+06	8.62E+07
*		

Figure 31. Tabular option

Besides this data, there are multiple inputs that apply for both approaches and are highly useful in design, but sadly not very applicable for this project. As it was explained before, there are many factors that should be consider while designing a real component; however, they are out of the scope of this study for the sake of simplicity and to focus on the core of the calculations. All these factors are explained, as it is important to understand what are all the possibilities that are provided by this tool, but not significantly relevant for the calculations in this project.

Fatigue Strength Factor (K_f):

This property represents the factor by which the fatigue strength is reduced, accounting for real-world environments that may be more severe than controlled laboratory conditions where the data was obtained. It allows adjustment of the stress-life or strain-life curves during the fatigue analysis. The possible values go from 0,01 to 1, being 1 the default. Even though it is called K_f, as the fatigue stress concentration factor that has been used during this project, it does not represent the same concept, in fact the fatigue stress concentration factor was previously obtained from the information of the stress calculation with FEA, while this fatigue strength factor is an input that gives the opportunity of adding another reducing factor, which could represent different conditions of reality that reduce the strength on the specimen.

Loading Type:

The Zero-Based ($R = 0$), Fully Reversed ($R = -1$), and Ratio options represent constant amplitude and proportional loading types, and these options are graphically illustrated in the Worksheet.

The History Data option allows you to import a file containing data points, representing a non-constant amplitude proportional loading type. The data is displayed in a graph on the Worksheet, where the number of data points to be plotted can be controlled using the Maximum Data Points To Plot property in the Options category.

The Non-proportional Loading option is suitable for non-proportional constant amplitude loading types where models alternate between two different stress states, such as bending and torsional loading. This feature can be used to model problems where alternating stress is imposed on a static stress. Non-proportional loading is supported only for Fatigue Tool objects used with Solution Combination, where exactly two stress states are selected.

Scale Factor:

This parameter enables the scaling of load magnitudes, for example, for a value of 3, the amplitude of a zero-based loading will be 1,5 times the stress in the body. This option is useful for observing the effects of different finite element loading magnitudes without the need to rerun the entire structural analysis. This scale factor is applied after the stresses have been collapsed from a tensor into a scalar. Therefore, any multiaxial stress collapse methods that are sensitive to sign (such as Von-Mises, Maximum Shear, Maximum Principal) may lead to different results if the scale factor had been applied directly to the environment load itself.

Analysis Type:

Definition of the fatigue analysis as either Stress Life or Strain Life

Mean Stress Theory:

This setting determines how mean stress effects should be handled. For the Stress Life approach, the options include None (default), Goodman, Soderberg, Gerber, ASME Elliptical, and Mean Stress Curves, while None, Morrow, and SWT (Smith-Watson-Topper) can be chosen in case of a Strain Life approach.

Method Selection:

In the case of Random Vibration analyses Narrow Band, Steinberg (default), and Wirsching can be selected.

Stress Component:

Since stresses are multiaxial while experimental fatigue data is usually uniaxial, the stress needs to be converted from a multiaxial stress state to a uniaxial one. A value of 2 times the maximum shear stress is commonly used. There are several options to choose from, including component stresses, von Mises (29), and signed von Mises, which considers the sign of the absolute maximum principal stress. The signed von Mises option is particularly useful for accounting for any compressive mean stresses.

$$\sigma_{eq} = \sqrt{\frac{1}{2}[(\sigma_{11}-\sigma_{22})^2+(\sigma_{22}-\sigma_{33})^2+(\sigma_{33}-\sigma_{11})^2]+3(\sigma_{12}^2+\sigma_{23}^2+\sigma_{13}^2)} \qquad (29)$$

Where σ_{eq} is the equivalent stress of von Mises, the $\sigma_{11}, \sigma_{22}, \sigma_{33}$ are the normal stresses and the $\sigma_{12}, \sigma_{23}, \sigma_{13}$ are the shear stresses.

Exposure Duration:

In the case of Random Vibration and Harmonic Response analysis, it specifies the duration (in time) for which the loading is applied, where the resulting damage is calculated for the entire duration.

Frequency Selection:

For Harmonic Response analyses, Single Frequency (default), Multiple Frequencies or Sine Sweep can be chosen.

Frequency:

This property is applicable only for Harmonic Response analysis. It specifies the frequency (in Hz) for which the stress response is calculated and used for the fatigue analysis, automatically determining the phase angle at which the maximum stress occurs for the chosen frequency.

Sweep Rate:

This property specifies the rate of frequency sweep in Hz/s units for Harmonic Response analysis and is available only when the Sine Sweep option is selected for the Frequency Selection property.

Units Name:

The Life Units can be specified as cycles, hours, blocks, days, seconds, months, minutes or user defined.

1 "Unit" is Equal To:

This field allows modifying the value based on the desired number of cycles or blocks for the selected Units Name. The term "unit" represents either cycle or block, depending on the Units Name selection.

Bin Size:

It determines the number of divisions the cycle counting history should be organised into for the history data loading type. The bin size, in a sense, specifies the dimensions of the rainflow matrix, where larger bin size provides greater precision but will increase solving time and memory usage.

Use Quick Rainflow Counting:

This option appears only if the Type is set to History Data (non-constant amplitude loading). By using a "quick counting" technique, runtime and memory usage can be significantly reduced, especially for long time histories.

Infinite Life:

In the case of stress life analysis, this option is visible only if the Type is set to History Data (non-constant amplitude loading) and determines the life to be used if the stress amplitude is lower than the lowest stress on the S-N curve. It is particularly important in assessing the damage caused by small stress amplitudes from the rainflow matrix.

On the other hand, in strain-life analysis, which is equation-based, there is no built-in limit as in stress-life analysis. By specifying Infinite Life, contour plots showing very high lives can be avoided. For example, setting a value of 10^9 cycles as the Infinite Life will result in a maximum reported life of10^9.

Maximum Data Points To Plot:

This option is applicable only for History Data loading and enables to specify the number of data points to be displayed in the corresponding graph in the Worksheet. The default value is 5000 points. While all data points are used in the analysis, setting a specific value helps avoid clutter and improve graph readability by displaying only selected points at regular intervals.

For the present case, not all the inputs are necessary or useful, the ones given were the S-N curve with respect to the mean stress, with the tabular option with a Log-Log interpolation mode, as shown on Figure 32.

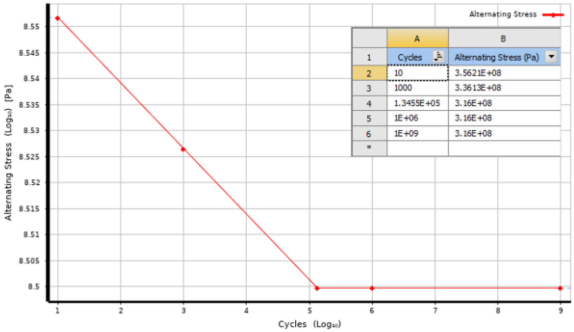


Figure 32: S-N curve

Outputs:

The fatigue tool provides the following five contour results, for which an example is presented, a "centre-hole" specimen with a notch radius of 0,5mm and an applied stress of 108MPa.

Life:

By using this option, the program displays the number of cycles of the part until the fatigue failure. It is represented as a colour map with a legend where the minimum and the maximum number of cycles are written (Figure 33). If the alternating stress is lower than the lowest defined in the S-N curve, the program provides the number of cycles of that point of the S-N curve as the Life result, i.e., infinite life, which in this case is 10^9. In the case of non-constant loading, it represents the number of blocks until failure, i.e., the time block of the given load history.

In summary, what the program does is to calculate the equivalent alternating stress, the von Mises equivalent for this case (Figure 37), for each node, compare it with the S-N curve and provide the number of cycles for each node's stress value.

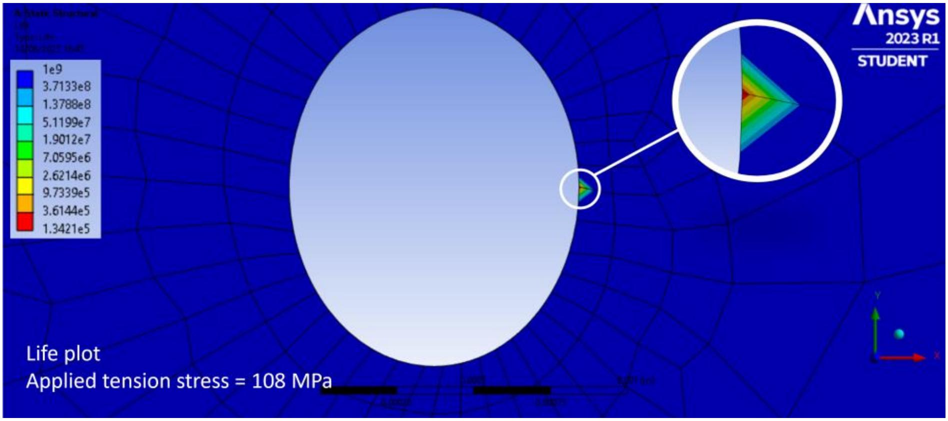


Figure 33: Fatigue Tool Life plot. Zoom of the circular notch

<u>Damage</u>:

This outcome provides the value of the design life divided by the available life, where the design life is 10^9 cycles by default and the available life is the result of life for each node. Thereby, the value of the damage will be 1 (there is no failure for that stress) or higher (there is failure).

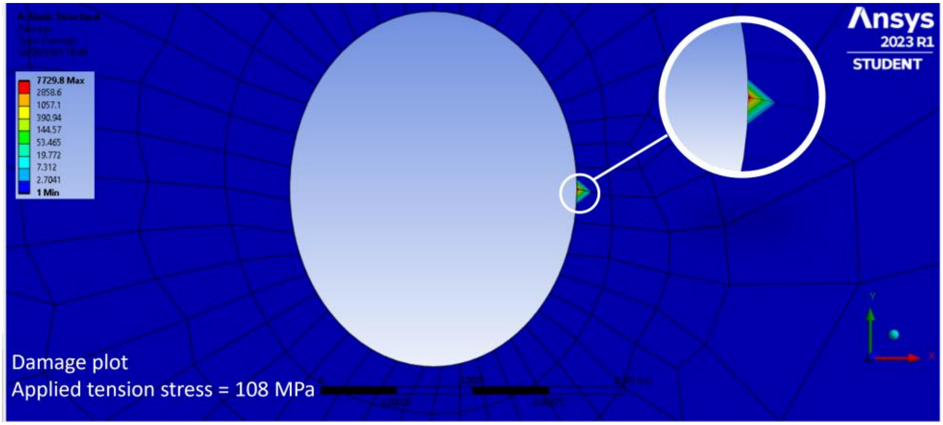


Figure 34: Fatigue Tool Damage plot. Zoom of the circular notch

Safety factor:

The depicted graph represents a contour plot of the factor of safety (FS) with respect to a fatigue failure at the design life. The maximum reported Safety Factor is 15.

1. Calculate the alternating and mean stress tensor.

2. Collapse alternating and mean stress from tensor to scalar using selected stress component.

3. Calculate Safety Factor from the mean stress equation using S_{eqv} as queried from the S-N curve for the design life

$$\frac{1}{FS} = \frac{S_{alt}}{S_{eqv}} + \frac{S_{mean}}{S_{ultimate}} \tag{30}$$

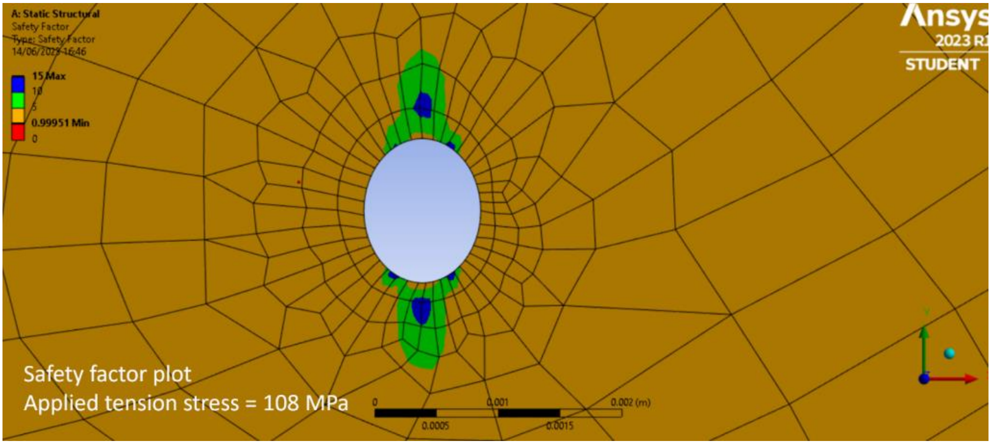


Figure 35: Fatigue Tool Safety Factor plot. Zoom of the circular notch

Biaxiality Indication:

This plot gives a qualitative measure of the stress state throughout the body, where -1 means pure shear, 0 means uniaxial stress and 1 means pure biaxial state.

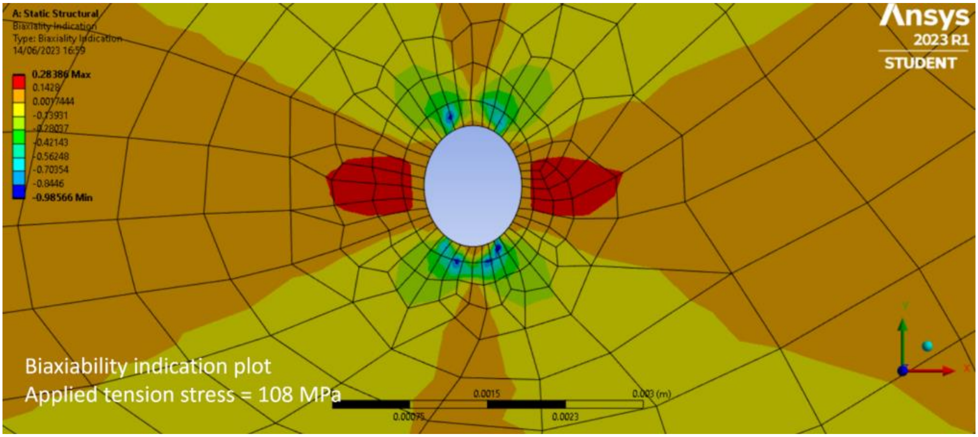


Figure 36: Fatigue Tool Biaxiality Indication plot. Zoom of the circular notch

Equivalent Alternating Stress:

The Equivalent Alternating Stress is determined by considering the effects of multiaxial loading and mean stress and is used to query the S-N curve. The following steps are taken at each node to calculate the Equivalent Alternating Stress:

1. Calculate the alternating and mean stress tensor.
2. Convert the alternating and mean stress from a tensor form to a scalar form using the selected stress component.
3. Calculate the Equivalent Alternating Stress using the specified empirical stress theory, as defined by the Mean Stress Theory property of the Fatigue Tool object. For example, if the Mean Stress Theory property is set to Goodman, the calculation of the Equivalent Alternating Stress follows the Goodman stress equation.

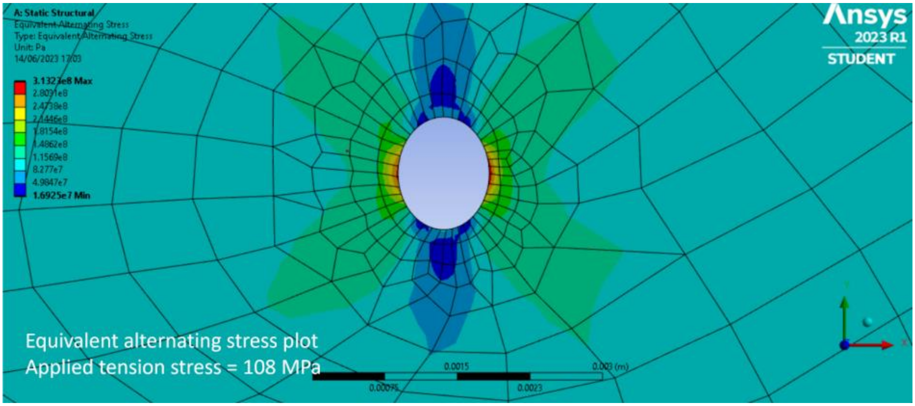


Figure 37: Fatigue Tool Equivalent Alternating Stress plot (equivalent of von Mises). Zoom of the circular notch

Besides the contour results, the fatigue tool also provides four graph results:

<u>Rainflow Matrix:</u>

The presented graph illustrates the distribution of cycle counts within each bin for history data problems. This information is reported at the specific scope point that exhibits the highest damage.

<u>Damage Matrix:</u>

Analogically to the rainflow matrix, this graph illustrates the proportional amount of damage caused by each bin for the history data option. This information provides insights into the accumulation of total damage, such as whether the damage resulted from numerous small stress reversals or a few significant ones.

<u>The Fatigue Sensitivity:</u>

The Fatigue Sensitivity plot depicts how the fatigue results vary based on the loading at the critical location within the scoped region. This sensitivity analysis can be conducted for parameters such as life, damage, or factor of safety. By setting the lower and upper fatigue sensitivity limits (e.g., 50% and 150% respectively) and applying a scale factor (e.g., 3), the plot will display data points along a scale ranging from 1,5 to 4,5. It is possible to customise the number of fill points in the curve and choose different chart viewing options, including linear or log-log.

Hysteresis:

Hysteresis is observed in a strain-life fatigue analysis when the local elastic/plastic response deviates from linearity, even though the finite element response remains linear. To account for this nonlinearity, the Neuber correction is employed to determine the local elastic/plastic response based on the linear elastic input. As a consequence of this nonlinear local response, repeated loading generates closely spaced hysteresis loops. In a constant amplitude analysis, only a single hysteresis loop is formed, whereas a non-constant amplitude analysis can generate multiple loops through rainflow counting.

The Hysteresis result displays the local elastic-plastic response at the critical location within the scoped result. Similar to other result items, the Hysteresis result can be scoped to focus on specific regions of interest. This visualisation aids in comprehending the actual local response, which may not be readily apparent. Notably, the Hysteresis result reveals that despite the loading/elastic result being tensile, the local response extends into the compressive region.

In summary, the fatigue tool enables the user to carry out multiple fatigue calculations and analyses without coding, facilitating the calculation of usual problems. However, this tool has an experimental try approach, i.e., the program does not choose a fatigue theory to address the problem or provide the fatigue limit of the specimen, but it confirms if the workpiece would or would not experience failure for the given conditions. For instance, the value of the K_f or K_t is not provided by the fatigue tool, hence the user has to calculate it using a theory self-selected.

6. COMPARISON OF RESULTS OBTAINED WITH EACH METHOD

A wide range of topics and information about Ansys has been provided so far in this thesis and it might seem like the presented work done with Ansys APDL is considerably far from what has been explained about Workbench. Thereby, this chapter aims to compare both followed procedures, as well as giving a better understanding about the possibilities with each interface.

The first important difference for this project is the approach used to calculate the fatigue strength or the failure of the component due to fatigue. For the simulations on APDL the Point Method of the Theory of Critical Distances was implemented, which is a significantly easy method part of Fracture Mechanics. Although it is not very complex, it is indeed highly adequate for many cases and especially for notches. Nevertheless, other methods can be also implemented on APDL, such as the S-N or the strain-life method, since the user is the responsible of coding the procedure in this language and with the provided library of functions, the right election of the approach and the accuracy of the results hinges on the knowledge of the user about materials physics and the user's skills to code properly.

On the other hand, Workbench presents only two possible approaches to choose from, stress-life and strain-life, which are the most basic approaches. As it was introduced in chapter 2, these methods are part of the first studies carried out on the field of failure and fatigue of materials. They are simple and definitely useful for many cases, but they are completely empirical and do not consider the existence of cracks nor other factors as the volume or the geometry might be. Also, the user only has to choose the approach and enter the asked information with no need of coding a calculation method, it is simpler in that regard but there is less freedom to implement a more adequate theory.

Regarding the inputs, a high number of them are available in Workbench (chapter 5) and they are highly useful for design of real components. These and more inputs can also be implemented in APDL, but it is significantly easier in workbench, as it does not have to be code, the function directly asks for the needed information.

However, the majority of these inputs do not meet the current study, as they refer to cases with variable amplitudes or external factors, very important in practice but not part of the basic calculation and its accuracy.

Focusing on the outputs, it happens something similar than with the inputs, there are very few possible outputs in Workbench and some are only useful for cases with variable amplitudes or frequencies or are just calculated based on the life output. In fact, from the possible outputs for this project, only the stresses (not part of the fatigue tool) and the life over the specimen are worth of analysis, since the damage and the safety factor are just calculated with the life data with simple operations, so their accuracy and information rely on the life output.

The data obtained from Ansys, as well as the procedure, is very different from the one in Workbench; nevertheless, the goal is to compare the accuracy of the failure prediction using both paths. In APDL the threshold was found with the help of the PM, i.e., below this value of stress, the specimen does not experience failure, while in Workbench this value cannot be asked for, but a stress is applied and the life output displays infinite life all over the workpiece or not. If the minimum life is lower than 10^9, the specimen might should experience failure in the hot spot. For this reason, to obtain the information with Workbench it is necessary to try different values of stress until finding the lowest one to induce failure. For this reason, the precision of the results relies on the precision of the user by trying different values. In this case it has a precision of 0,1MPa, since the specimen was showed to have infinite life for 107,9MPa. Table 3 aims to present these results in the clearest way possible.

Table 3: Comparison of results obtained with APDL and Workbench

	Ansys	Workbench	
	Failure Threshold	Applied Stress	Minimum Life
Centre Hole	148,34MPa	108MPa	129370N
V-notch	51,31MPa	38MPa	62931N

Based on these results, it is reasonable to state that Workbench provides more conservative results, and by a wide difference. Looking back to section 2.4 The

Failure of Notched Specimens, it was explained that notched specimens can bear higher loads than the theoretical ones, i.e., the fatigue limit of the workpiece should be the material fatigue limit divided by the theoretical stress concentration factor (K_t), but, in reality, the fatigue limit is less decreased, being divided by the stress concentration factor on the critical distance (K_f). Therefore, the solution obtained by Workbench is significantly more conservative and thus less efficient, a piece designed using the fatigue tool would be working under non optimal conditions, even though the designer would think the opposite. This lack of accuracy relies on the fact that this tool is only considering the stress-life curve, leaving apart other important factors which affect reality.

7. CONCLUSIONS

As shown in the previous section, the fatigue tool of Workbench is highly more conservative and less accurate than the calculations carried out with APDL. This problem can also be seen in the accuracy of the meshing, the mesh automatically created by Workbench is not fine enough around the notch and it is necessary to insert refines around that area by hand. Nonetheless, the possibilities to refine are not many, therefore, the user cannot apply this refine as exactly as wanted or in the right spot.

On the other hand, APDL is more difficult to learn by a new user, as it is a coding language and not an interface with options to choose. However, after the learning phase, APDL offers a wider range of possibilities in the field of fatigue, especially for a designer, as it provides the freedom to the user of writing every detail of the problem, conditions, solution, theory, etc. Besides that, APDL facilitates small changes of the problem or basing new problems on old ones by rewriting some lines of the code, while in Workbench it is necessary to create a new project.

One of the best features of the fatigue tool is the numerous possible inputs, mostly for cases with variable amplitude or random vibrations, since these cases are significantly difficult to code and Workbench offers many options which represent usual problems.

In my opinion, the fatigue tool is highly powerful for beginners and representations, since it is a more visual interface, the plots have a better visual quality and it is simpler for a user without a solid knowledge in fatigue of materials, fracture mechanics or coding, but this could also be dangerous, since an inexperience user would not notice many mistakes or inaccuracies. However, for deeper purposes, such as academic, design, optimisation, research, etc., APDL provides a higher accuracy and freedom for a user with the required background.

Now focusing on the implementation of the Theory of Critical Distances to APDL, the results are certainly accurate if compared to empirical solutions, which shows the high potential of this theory and its simple implementation to this software.

To finalise, this study presents the difficulties of atomising fatigue calculations. Fatigue of materials is a complicated field of physics and engineering, where researchers could not find a final solution yet, but a pool of tools, theories and approximations that represent increasingly better the reality, but that entails the need for a thinking mind with the appropriate expertise to follow the right path and obtain optimal solutions.

8. CONCLUSIONES

Como se ha mostrado en la sección anterior, la herramienta de fatiga de Workbench es mucho más conservadora y menos precisa que los cálculos realizados con APDL. Este problema también se puede ver en la precisión del mallado, la malla creada automáticamente por Workbench no es lo suficientemente fina alrededor de la muesca y es necesario insertar refinados alrededor de esa zona a mano. Sin embargo, las posibilidades de refinado no son muchas, por lo que el usuario no puede aplicar este refinado con la exactitud deseada o en el punto correcto.

Por otra parte, APDL es más difícil de aprender para un nuevo usuario, ya que se trata de un lenguaje de codificación y no de una interfaz con opciones para elegir. Sin embargo, después de la fase de aprendizaje, APDL ofrece una gama más amplia de posibilidades en el campo de la fatiga, especialmente para un diseñador, ya que proporciona la libertad al usuario de escribir cada detalle del problema, condiciones, solución, teoría, etc. Además, APDL facilita pequeños cambios del problema o basar nuevos problemas en antiguos reescribiendo algunas líneas del código, mientras que en Workbench es necesario crear un nuevo proyecto desde el principio.

Una de las mejores características de la herramienta de fatiga son las numerosas entradas posibles, sobre todo para casos con amplitud variable o vibraciones aleatorias, ya que estos casos son significativamente difíciles de programar y Workbench ofrece muchas opciones que representan problemas habituales.

En mi opinión, la herramienta de fatiga es muy potente para principiantes y representaciones, ya que es una interfaz más visual, los gráficos tienen una mejor calidad visual y es más sencilla para un usuario sin conocimientos sólidos en fatiga de materiales, mecánica de fractura o programación, pero esto también podría ser peligroso, ya que un usuario inexperto no notaría muchos errores o imprecisiones. Sin embargo, para fines más profundos, tales como académicos, diseño, optimización, investigación, etc., APDL proporciona una mayor precisión y libertad para un usuario con la formación requerida.

En cuanto a la aplicación de la Teoría de las Distancias Críticas a APDL, los resultados son ciertamente precisos si se comparan con las soluciones empíricas, lo que demuestra el gran potencial de esta teoría y su sencilla aplicación a este software. La fatiga de materiales es un campo complicado de la física y la ingeniería, en el que los investigadores aún no han podido encontrar una solución definitiva, sino un conjunto de herramientas, teorías y aproximaciones que representan cada vez mejor la realidad, pero que conlleva la necesidad de una mente pensante con los conocimientos adecuados para seguir el camino correcto y obtener soluciones óptimas.

REFERENCES

[1] (2023) *Ansys Mechanical User's Guide*

[2] C. Bathias, A. Pineau (2008) *Fatigue of Materials and Structures* – Editorial: Wiley-ISTE

[3] V. Chaves, A. Navarro, C. Madrigal. (2015) *Stage I crack directions under in-phase axial–torsion fatigue loading for AISI 304L stainless steel.* - International Journal of Fatigue 80 (2015) 10–21

[4] V. Chaves, A. Navarro, C. Madrigal. (2016) *Fatigue limit predictions at stress concentrations using FEA and microstructural fracture mechanics.* - Theoretical and Applied Fracture Mechanics 87 (2017) 11–20

[5] D. Gaia da Silva, J.T. Lockwood, W. Liang, T.H. Topper (2021) *Mean stress effect in stress-life for hard steels* - International Journal of Fatigue 146, 106101

[6] Y. Lee, J. Pan, R Hathaway, M. Barkey (2005) *Fatigue Testing and Analysis* – Editorial: Elsevier Science Ltd

[7] Y.B. Liu, Y.D. Li, S.X. Li, Z.G. Yang, S.M. Chen, W.J. Hui, Y.Q. Weng (2010) *Prediction of the S–N curves of high-strength steels in the very high cycle fatigue regime* - International Journal of Fatigue 32 (2010) 1351–1357 – Pag. 1352-1354

[8] S. Müftü (2022) *Finite Element Method* – Editorial: Elsevier Science Ltd

[9] A. Navarro, V. Chaves, J.A. Balbín (2021) *Variations on a critical distance theme* - International Journal of Fatigue 152 (2021) 106453

[10] D. Roylance (2001) *Stress-Strain Curve* – Lecture notes, Massachusetts Institute of Technology Cambridge, MA 02139

[11] D. Roylance (2001) *Introduction to Fracture Mechanics* - Lecture notes, Massachusetts Institute of Technology Cambridge, MA 02139

[12] D. Roylance (2001) *Finite Element Analysis* - Lecture notes, Massachusetts Institute of Technology

[13] S. Suresh. (1991). *Fatigue of Materials* – Editorial: Cambridge University Press

[14] D. Taylor. (2007). *The Theory of Critical Distances* – Editorial: Elsevier Science Ltd

[15] M. Vidal (2021) *Trabajo de fin de máster: Fatiga aeronáutica, dimensionamiento y análisis a fatiga del cajón de encastre del Boeing 737-400*

[16] A. T. Zehnder (2012) *Fracture Mechanics* – Editorial: Springer Cambridge, MA 02139

APPENDIX: APDL CODES

Centre Hole Code (example for a radius of value 0,5):

```
FINISH $ /CLEAR


! Dimensions

W=500                                    ! Wide [mm]

H  =1000                                 ! Height [mm]

pi=3.14159265

r=0.5                                    ! Radius changing parameter [mm]

W_min=W-(2*r)                            ! depends on the radius [mm]

S_e=1                                    ! Efective stress [MPa]

S_a=S_e*(W_min)/(W)     !Applied stress S_a < S_e [MPa]

K_th= 15.03                              ! Fatigue Threshold [MPa*(m)^(1/2)]

Sigma_fl= 632                            ! Ultimate Tensile Stress [MPa]

L= 1000*((K_th**2)/(Sigma_fl**2))/pi ! Taylor's length


/prep7

! Definition of the material and the specimen

MP,EX,1,200000 !Young's module [MPa]

MP,NUXY,1,0.3 ! Friction coefficient

ET,1,PLANE183,,,0 !2D element of 8 nodes, plane stress
```

```
! Keypoints of the geometry
K,1,0,0
K,2,W_min/2,0
K,3,W/2,r
K,4,W/2,H/2
K,5,0,H/2
K,6,W/2,0

! Lines
L,1,2
LARC,2,3,6,r ! Circle arc
L,3,4
L,4,5
L,5,1
AL,ALL ! Area

! Meshing features
LESIZE,1,,,200,0.05
LESIZE,2,,,30
LESIZE,3,,,400,20
LESIZE,4,,,10
LESIZE,5,,,20
AMESH,ALL ! Meshing
KREFINE,2,,,,
KREFINE,2,,,,
```

```
KREFINE,3,,,,

LREFINE,2,,,,


! Boundary Conditions

DL,1,1,SYMM ! Symmetry on line 1

DL,3,1,SYMM ! Symmetry on line 3

SFL,4,PRES,-S_a,  ! Pressure stress, the minus turns it into a tensile stress


/SOLU $ SOLVE ! Solution

! Analysis of results

/POST1

PLNSOL,S,Y,   ! Representation of stresses

PLNSOL,U,SUM,2      ! Representation of displacements


!Path with the stress for the Point Method

PATH,bisector,2,,100

PPATH,1,,W_min/2,0,0 $ PPATH,2,,(W_min/2)-L,0,0,

PDEF,1,S,Y,

PRPATH,1

PLPATH,1


! Getting scalar values

*GET,Smaxima,PATH,0,MAX,1  !Maximum stress

*GET,STaylor,PATH,0,ITEM,1,PATHPT,51 ! Stress on L/2, i.e., K_f. Point number 51 of the path
which is 100 points in total.
```

V-Notch Code (example for a radius of value 0,5)

```
FINISH $ /CLEAR
! V notch
! Dimensions
W=500                           ! Wide [mm]
H  =1000                         ! Height [mm]
pi=3.14159265
phi=30*pi/180                    ! Angle of the notch [rad] (changing parameter)?
rho=0.5                         ! Radius of the notch [mm] (changing parameter)
d=5+rho                            ! Length of the notch [mm] (changing parameter)?
W_min=W-(2*d)                    ! depends on the length of the notch [mm]
S_e=1                            ! Efective stress [MPa*(m)^(1/2)]
S_a=S_e*(W_min)/(W)    !Applied stress S_a < S_e [MPa*(m)^(1/2)]
K_th= 15.03                        ! Fatigue Threshold [MPa*(m)^(1/2)]
Sigma_fl= 632                    ! Ultimate Tensile Stress [MPa]
L= 1000*((K_th**2)/(Sigma_fl**2))/pi ! Taylor's length

! Helping parameters
alpha=pi/2-phi                    ! [rad]
a= rho*sin(alpha)/sin(phi)        ! [mm]
b=(d-rho*(1-cos(alpha)))/cos(phi)  ! [mm]

/prep7
! Definition of the material and the specimen
MP,EX,1,200000 !Young's module [MPa]
MP,NUXY,1,0.3   ! Friction coefficient
ET,1,PLANE183,,,0 !2D element of 8 nodes, plane stress

! Keypoints of the geometry
```

```
K,1,0,0
K,2,W/2,0
K,3,W/2,H/2
K,4,0,H/2
K,5,0,(a+b)*sin(phi)
K,6,b*cos(phi),a*sin(phi)
K,7,d,0
K,8,d-rho,0

! Lines
L,7,2
L,2,3
L,3,4
L,4,5
L,5,6
LARC,6,7,8,rho ! Circle arc
AL,ALL ! Area

! Meshing features
LESIZE,1,,,200,30
LESIZE,2,,,20
LESIZE,3,,,20
LESIZE,4,,,400,0.05
LESIZE,5,,,50
LESIZE,6,,,50
AMESH,ALL ! Meshing
KREFINE,7,,,,
KREFINE,7,,,,
LREFINE,6,,,,
```

```
! Boundary Conditions
DL,1,1,SYMM ! Symmetry on line 1
DL,2,1,SYMM ! Symmetry on line 3
SFL,3,PRES,-S_a,  ! Pressure stress, the minus turns it into a tensile stress

/SOLU $ SOLVE ! Solution

! Analysis of results
/POST1
PLNSOL,S,Y,   ! Representation of stresses
PLNSOL,U,SUM,2       ! Representation of displacements

!Path with the stress for the Point Method
PATH,bisector,2,,100
PPATH,1,,d,0,0 $ PPATH,2,,(d+L),0,0,
PDEF,1,S,Y,
PRPATH,1
PLPATH,1

! Getting scalar values
*GET,Smaxima,PATH,0,MAX,1  ! Maximum stress
*GET,STaylor,PATH,0,ITEM,1,PATHPT,51 ! Stress in L/2, point number 51 of the path which is 100 points
in total.
```